UNLIMITED DISTRIBUTION

National Defence
Research and
Development Branch

Défense nationale
Bureau de recherche
et développement

DREA CR/92/454

PROPELLER VIBRATION AND STRESS ANALYSIS BY FINITE ELEMENT METHODS, PVAST VERSION 4
USER'S MANUAL

by
M.W. Chernuka - M.E. Norwood - T.S. Koko

MARTEC Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada
B3J 3J8

Scientific Authority
D.C. Stredulinsky

W7707-0-1091/01-OSC
Contract Number

July 1992

CONTRACTOR REPORT

Prepared for

Defence
Research
Establishment
Atlantic

Centre de
Recherches pour la
Défense
Atlantique

Canada

ABSTRACT

A User's Manual for a computerized procedure for propeller vibrations and stress analysis by finite element methods is presented.

The manual provides a general description of the procedure, including the analysis capabilities, the numerical techniques used to formulate and solve the equations of motion, modelling techniques, and computer program organization and operation.

RÉSUMÉ

Un guide d'utilisation d'une méthode informatisée d'analyse par éléments finis des vibrations et des contraintes dans les hélices est présenté.

Le guide présente une description générale de la méthode, y compris des possibilités d'analyse, les techniques numériques utilisées pour formuler et résoudre les équations de mouvement, des techniques de modélisation, et l'organisation et le fonctionnement du programme informatique.

ACKNOWLEDGEMENT

The computer program described in this manual has been developed for the Defence Research Establishment Atlantic under the following Canadian Department of National Defence sponsored industrial research contracts:

SEY.7040073 Serial No. 2EY3-0365
SEY.7040124 Serial No. 8EY5-0010
4EZ.1431514 Serial No. 8EZ76-00063
OAD.3217050 Serial No. 2AD78-00033
OAD.3219053 Serial No. 2AD80-00058
10SC.97707-2-3651 Serial No. 8SC82-00313
10SC.97707-4-6175 Serial No. 8SC84-00422
10SC.97707-5-6305 Serial No. 8SC85-00338
OSC87-00039-(010) Serial No. W7707-7-7855/01-OSC
OSC90-00177-(014) Serial No. W7707-0-1091/01-OSC

TABLE OF CONTENTS

Page

Title Page i
Abstract ii
Acknowledgements iii
Table of Contents iv
Summary of Appendices vi

1.0 INTRODUCTION 1.1

2.0 THEORETICAL FORMULATION 2.1
- 2.1 Equations of Motion 2.1
  - 2.1.1 Element to Structure Matrices 2.1
  - 2.1.2 Hydrodynamic Added Mass 2.2
  - 2.1.3 Boundary Conditions 2.3
  - 2.1.4 Geometric Stiffness 2.4
- 2.2 Static Analysis 2.4
- 2.3 Calculation of Natural Frequencies and Mode Shapes 2.5
- 2.4 Dynamic Analysis 2.5
  - 2.4.1 Time History Analysis 2.5
  - 2.4.2 Response Spectrum Analysis 2.7
  - 2.4.3 Frequency Response Analysis 2.8
- 2.5 Substructures 2.8
- 2.6 Numerical Error Estimation 2.8

3.0 FINITE ELEMENT LIBRARY 3.1
- 3.1 Structural Elements 3.1
  - 3.1.1 Thick/Thin Shell Element 3.1
  - 3.1.2 Three-Dimensional Solid Element 3.3
  - 3.1.3 Tetrahedron Element 3.3
  - 3.1.4 Transition Element 3.4
  - 3.1.5 Triangular Membrane Element 3.4
  - 3.1.6 Rod Element 3.4
  - 3.1.7 Gap Element 3.4
- 3.2 Fluid Elements 3.5

4.0 PROPELLER MODELLING 4.1
- 4.1 Introduction 4.1
- 4.2 Structural Models 4.1
  - 4.2.1 Geometry Considerations 4.1
  - 4.2.2 Modelling Options 4.1
  - 4.2.3 Attachment Bolts 4.2
  - 4.2.4 Hub Keyway Model 4.3
  - 4.2.5 Air Channel Model 4.3

TABLE OF CONTENTS (Continued)

Page

4.3 Modelling Surrounding Fluid 4.3
4.3.1 Fluid Elements 4.3
4.3.2 Surface Panel Method 4.4

5.0 PROGRAM ORGANIZATION AND OPERATION 5.1
5.1 General 5.1
5.2 Input/Output 5.1

6.0 INPUT PREPARATION USING PRE-PROCESSING PROGRAMS 6.1
6.1 Introduction 6.1
6.2 Structural Finite Element Models 6.1
6.3 Hydrodynamic Added Mass Finite Element Model 6.6
6.4 Load Generation 6.7

7.0 INTERPRETATION AND PRESENTATION OF RESULTS USING POST-PROCESSING PROGRAMS 7.1

8.0 GUIDELINES ON HOW TO USE PVAST EFFECTIVELY AND EFFICIENTLY 8.1

9.0 REFERENCES 9.1

SUMMARY OF APPENDICES

Page

A. INPUT DATA FOR PROGRAM VAST A.1

B. INPUT DATA FOR GENERATING BLADE AND ROOT MODELS B.1
  - B.1 PVBLD1: Computation of the true space coordinates for propeller blades using the designer's basic blade design data B.1
  - B.2 PVFIL, PVHUB, PVPALM: Generation of finite element models for fillet and propeller palm or hub B.7
  - B.3 PVBLD2: Generation of finite element model for propeller blade B.21
  - B.4 PVBLD3: Generation of blade model using a graphics tablet B.24
  - B.5 PVSUB1, PVSUB2: Generation of substructured finite element model of combined blade, fillet, hub or palm and optionally bolts as rods or bolts and bolthole substructures modelled in detail B.28
  - B.6 PVKEY: Generation of two dimensional finite element models for hubs with keyways and their loads B.31
  - B.7A PVBCON: Interactive definition and/or modification of boundary conditions for finite element model B.34
  - B.7B PVGAP: Consolidation into a single file of all gap element information for the finite element model defined automatically by programs or optionally defined interactively by the user in earlier program steps but generally stored on several independent data files with provisions for final interactive modification B.37
  - B.8 PVBLTA/B/C: Attachment bolt and gap element modelling for palm models B.41
    - B.8A PVBLTA: Plotting the propeller palm base for the interactive relocation of nodes to achieve suitable bolt attachment points and to identify nodes for gap element attachments or boundary condition specifications B.43
    - B.8B PVBLTB: Plotting the propeller palm base for the interactive relocation of nodes to achieve suitably shaped elements for replacement with bolthole substructures and connecting bolt substructures Provisions are also made to select nodes on the propeller palm base to which gap elements will be attached or boundary conditions applied B.48
    - B.8C PVBLTC: Plotting the base of bolthole substructures for identification of nodes for gap elements and boundary conditions B.54

SUMMARY OF APPENDICES (Continued)

Page

C. GENERATION OF LOADS C.1
- C.1 PVLODA: Adjusts applied propeller pressure loading to reduce pressure spikes along the leading and trailing edges from which loading errors develop C.1
- C.2 PVLODB: Transforms static or dynamic propeller blade pressure data into form acceptable to PVLODV C.3
- C.3 PVLODV: Generates load data for the PVAST finite element model of the propeller blade C.6

D. INPUT DATA FOR POST-PROCESSING PROGRAMS D.1
- D.1 PVDS1: Processes results from static analysis for subsequent plotting by PGDIS2/3 and PGSTR1/2/3 ..... D.1
- D.2 PVDS2: Processes results from dynamic analysis for subsequent plotting by PGDS1 ..... D.2
- D.3 PVFOR1: Processes force vector data for plotting by PGFOR1 ..... D.3
- D.4 PVSECT: Interactive PVAST post processing program for the identification of element surfaces on either horizontal or vertical propeller cross-sections for stress contouring purposes. PVSECT identifies vertical surfaces in the root (if it exists) as well as the blade but horizontal surfaces in the blade only. File created serves as input to POSTV2 ..... D.4
- D.5 PVFOR2: Computes from the results of a PVAST dynamic analysis for a single propeller blade (operating in a wake), the time histories of force and moment resultants acting on the shaft for a single blade or for the whole (multi-bladed) propeller D.6

E. OPERATING INSTRUCTION FOR PLOTTING PROGRAMS E.1
- E.1 PGBLAD: Plots expanded outlines, developed sections, developed outlines, transverse projections, longitudinal projections, plan views and grid proportions for the blade finite element model E.1
- E.2A PGLOD1: Plots input and adjusted pressure loading chordwise distributions as well as time histories for dynamic pressure loading E.3
- E.2B PGLOD2: Plots static or dynamic loadings E.5
- E.3 PGFEM: Plots finite element grids for blade and palm/hub models E.8
- E.4 PGDIS1: Plots deflected blade shapes resulting from static loading and/or natural mode shapes from a free vibration analysis E.9
- E.5 PGDS1: Provides displacement and stress time histories E.10
- E.6 PGSTR1: Plots stresses for static analysis E.12
- E.7 PGDIS2: Plots displacements for static analysis E.13

SUMMARY OF APPENDICES (Continued)

Page

E.8 PGDIS3: Plots loaded blade sections superimposed on unloaded sections E.14
E.9 PGSTR2: Plots principal stress contours for static analysis E.15
E.10 PGDIS4: Plots natural mode shapes superimposed on unloaded blade section E.16
E.11 PGDIS5: Plots contour lines of mode shapes E.17
E.12 PGSTR3: Plots principal stress vectors for static analysis E.18
E.13 PGFOR1: Plots radial, tangential and normal force vectors for either static or dynamic analysis E.19
E.14 PGLOD3: Plots time histories of input pressures, finite element pressures and drag forces E.21
E.15 PGFOR2: Plots time history curves of resultant propeller forces and moments acting on the shaft for a single or multi-blade propeller E.23

F. GENERATION OF A FINITE ELEMENT OR A SURFACE PANEL MODEL FOR THE FLUID SURROUNDING THE BLADE F.1
F.1 Introduction F.1
F.2 PVFL1/PVFLU2/PVFLU3: Input data for generation of fluid finite element model F.2

G. OPERATING INSTRUCTIONS FOR PLOTTING PROGRAM PGFLUD FOR PLOTTING FLUID FINITE ELEMENT MODELS G.1

H. OPERATING INSTRUCTIONS FOR PLOTHD (Hidden Line Removal) H.1

I. BLADE GEOMETRY GENERATION I.1

J. OPERATING INSTRUCTIONS FOR SUBPROGRAM PVUSE J.1

K. PRE-PROCESSING PROGRAM PVAST4 ..... K.1

L. POST-PROCESSING PROGRAM PPOST4 ..... L.1

M. THE GEOMETRY OF MARINE PROPELLERS M.1

N. GENERATION OF AIR CHANNEL MODELS N.1

1.0 INTRODUCTION

PVAST is a computer program for the strength and vibration analysis of marine propellers. It is based on the finite element method of structural analysis and is designed to automatically generate finite element models from a large library of elements for a wide variety of propeller geometries. The program can graphically display the geometry and the finite element model for data checking purposes. Static, natural frequency and dynamic response analyses can be carried out with the program. The analytical results can be processed and displayed in graphic form for ease of interpretation to show stresses, displacements, natural frequency of vibration and dynamic response.

The program is composed of three modules: one of interactive pre-processing programs, one of analysis programs and one of post-processing programs, see Figure 1.1. The pre-processing module is compatible with the 4010 series Tektronix graphics terminals and plotters and is used for data input verification, generation and graphic display of finite element models, and for the description and display of loads and boundary conditions. Terminal screen messages prompt the user in the selection of the pre-processing options to display the desired graphics. Files generated by the pre-processing programs are stored on disk or tape for use by the analysis programs. The routing through the analysis module (VAST) is determined by user decisions entered by terminal input. On completion of the analysis the post-processing module is loaded. The programs in this module are highly interactive and are used to convert the analytical results into a variety of graphic forms as desired by the user. Stress contour plots, vibration mode shapes and time histories of displacement and stress are some of the post-processing options available.

Brief descriptions of the finite element method, the formulation of the equations of motion, and the matrix equation solving schemes used in performing static and dynamic analyses and extraction of natural frequencies are given in Section 2. The static analysis capability includes the computation of structural deformations and element stresses caused by centrifugal forces, pressure loads, concentrated loads and/or prescribed displacements. The dynamic analysis capability includes the computation of structural deformations and element stresses for time varying pressure loads, arbitrary nodal applied loads, and support motion. Natural frequencies and mode shapes of propellers vibrating in air or water can be computed. In addition, the user has the option to include the effect of geometric stiffness to take into account the effect of centrifugal stiffening on natural frequency.

The finite elements offered for propeller modelling and modelling options available are described in Sections 3 and 4, respectively. Program organization and operation for PVAST are described in Section 5. Details on the features and usage of the program PVAST for pre- and post-processing are provided in Sections 6 and 7. Guidelines on how to use PVAST effectively and efficiently are provided in Section 8.

It is important that the user of PVAST have a basic understanding of propeller geometry. This is particularly true when using the pre-processing programs. A detailed description of the geometry of marine propellers is provided in Appendix M of this manual. Users with little or no background in propeller geometry should study Appendix M in detail. Users well knowledged in propeller geometry should read Appendix M to become familiar with such aspects as coordinate systems, sign-conventions, and terminology.

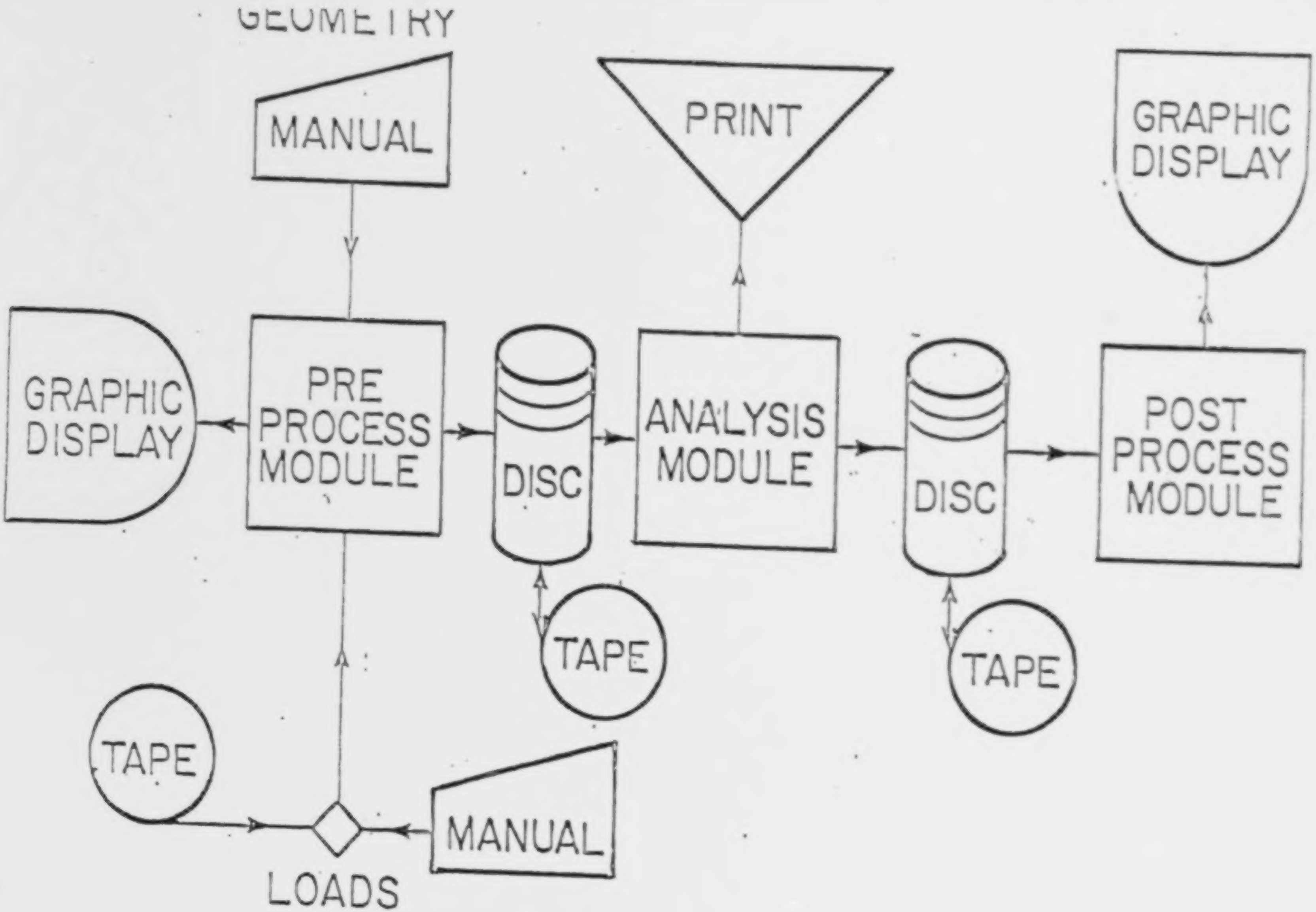

GEOMETRY
MANUAL
PRINT
GRAPHIC DISPLAY
GRAPHIC DISPLAY
PRE PROCESS MODULE
DISC
ANALYSIS MODULE
DISC
POST PROCESS MODULE
TAPE
TAPE
TAPE
MANUAL
LOADS

FIGURE 1.1: PVAST Modules

2.0 *THEORETICAL FORMULATION*

2.1 Equations of Motion

2.1.1 Element to Structure Matrices

The basis of the finite element method is the representation of a continuous structure by discretization, whereby the structure is divided into an equivalent system of smaller units, called elements, interconnected at a discrete number of points called nodes.

The nodal point equilibrium equations for the discretized system of structural elements can be expressed in matrix notation as follows:

$$[M]\{\ddot{\delta}\} + [C]\{\dot{\delta}\} + [K]\{\delta\} = \{f(t)\} \tag{2.1}$$

where [M] is the mass matrix, [C] is the damping matrix, and [K] is the stiffness matrix. The vectors $\{\delta\}$, $\{\dot{\delta}\}$, $\{\ddot{\delta}\}$, and $\{f(t)\}$ are the nodal displacements, velocities, accelerations, and loads, respectively.

The structure mass and stiffness matrices are formed by direct addition of the element matrices:

$$[M] = \sum_{e=1}^{NSE} [M]_e \,; \quad [K] = \sum_{e=1}^{NSE} [K]_e \tag{2.2}$$

where $[M]_e$ and $[K]_e$ are the mass and stiffness matrices, respectively, for the e'th element, and NSE is the total number of structural elements. The element mass and stiffness matrices each consist of a set of coefficients derived from the material and geometric properties of the element and obtained by use of variational principles.

The damping is assumed to be proportional to the velocity and is specified in the form of a modal critical damping ratio.

The load vector $\{f(t)\}$ is formed by using either the consistent or lumped approach. The latter is straight forward, the loads being specified at selected degrees-of-freedom. When the consistent approach is used, however, the load vector $\{f(t)\}$ is formed by direct addition of the element vectors:

$$\{f(t)\} = \sum_{e=1}^{NSE} \{f(t)\}_e \tag{2.3}$$

where $\{f(t)\}_e$ is the consistent load vector for the e'th element, again derived by use of variational principles.

2.1.2 Hydrodynamic Added Mass

The interaction between a submerged vibrating structure and its surrounding fluid results in the motion of the fluid and an induced pressure loading at the interface. The motion of the fluid by way of the induced interface pressures may be thought of as contributing to the mass of the structure.

The equation of motion for a submerged structure can be written as follows:

$$([M] + [M_A])\ \{\ddot{\delta}\} + [C]\ \{\dot{\delta}\} + [K]\ \{\delta\} = \{f(t)\} \tag{2.4}$$

where $[M_A]$ is called the "added mass" matrix.

A detailed description of the theoretical formulation of the added mass matrix is given in Reference [2]. The finite element method is used. The added mass matrix $[M_A]$ is calculated via the following equation:

$$[M_A] = \rho [F]^T\ [H^*]^{-1}\ [F] \tag{2.5}$$

where ρ is the water density, $[F]$ is the assembled interface matrix, and $[H^*]$ is the reduced fluid matrix. The interface matrix $[F]$ and the unreduced fluid matrix $[H]$ are formed by direct addition of the interface and fluid element matrices:

$$[F] = \sum_{e=1}^{NIE} [F]_e \; ; \quad [H] = \sum_{e=1}^{NFE} [H]_e \tag{2.6}$$

where $[F]_e$ is for the e'th interface element; $[H]_e$ is for the e'th fluid element, NIE is the total number of interface elements, and NFE is the total number of fluid elements.

2.1.3 Boundary Conditions

The structural problem described by Equation (2.1) is not completely specified unless boundary conditions are prescribed. Physically, if some sort of support or kinematic constraints are not imposed, a loaded structure is free to experience unlimited rigid body motion. Mathematically if this condition exists, the stiffness matrix [K] is singular, i.e. its determinant vanishes and its inverse does not exist.

Displacement boundary conditions are accounted for by making appropriate modifications to the equations of motion. If a displacement component is to be prescribed, say Δ_i at degree-of-freedom i, the equation

$$K\Delta_i = f_i \tag{2.7}$$

is added to Equation (2.1). Here $K \gg K_{ii}$ (where K_{ii} is the term on the diagonal of [K], corresponding to degree-of-freedom i) and f_i is a lumped load applied at degree-of-freedom i. This, in effect, links the degree-of-freedom i to a completely rigid base via a very stiff spring. The prescribed displacement Δ_i is approximately equal to f_i/K. A boundary condition of zero displacement is prescribed when f_i is set equal to zero (the program described in this manual uses a default value of $K=10^{22}$).

Whereas the displacements are the degrees-of-freedom associated with the structural problem, the velocity potential is the degree-of-freedom associated with the fluid or added mass problem. At restricted fluid boundaries, the condition of zero velocity potential must be imposed. Zero velocity potential must be specified at free surfaces as well. This, in effect, assumes no surface waves, which is valid in low frequency ranges. The program described in this manual incorporates boundary conditions of zero velocity potential simply by eliminating corresponding rows and columns from the interface and fluid element matrices.

2.1.4 Geometric Stiffness

The effect of the geometric stiffness is to stiffen a structure due to an initial loading resulting from centrifugal forces. This "centrifugal stiffening" is due to the additional strain energy resulting from nonlinear strain components which are not accounted for in the standard stiffness matrix $[K]_e$.

Element geometric stiffness matrices are derived from geometric properties and initial stresses due to centrifugal forces. Like the overall mass and stiffness matrices, the overall geometric stiffness matrix is formed by direct addition of the element matrices.

In order to determine the initial stresses required to formulate the geometric stiffness, a static analysis for the case of centrifugal loading must first be performed.

2.2 Static Analysis

The static analysis option for PVAST can be used to solve for the displacements and stresses in a structure subjected to forces and prescribed displacements.

The governing equations for a static analysis are:

$$[K]\{\delta\} = \{f\}$$

where [K] is the assembled stiffness matrix; $\{\delta\}$ is the displacement vector and {f} is the applied load vector. The equations are solved for the displacements (by decomposition), and these displacements are then used to calculate stresses.

PVAST offers the following types of static loading:

(a) concentrated point loads (at node points and/or anywhere within elements);

(b) pressure (distributed element loads);

(c) inertia forces due to translational acceleration (self-weight included), rotational velocities and rotational accelerations;

(d) constant acceleration support motion.

Types (a), (b) and (c) can be combined; type (d) must be considered separately.

2.3 Calculation of Natural Frequencies and Mode Shapes

The governing equations for a natural frequency analysis are [2]:

$$[\omega^2 ([M] + [M_A]) - ([K] + [K_G])] \{A\} = 0$$

where [M] is the structural mass matrix; $[M_A]$ is the added mass matrix (to account for fluid effects); [K] is the stiffness matrix; $[K_G]$ is a geometric stiffness matrix to account for preloading effects (i.e. centrifugal stiffening); x is a natural frequency; and {A} is the corresponding mode shape. The geometric stiffness matrix is formed using preloading either provided by the user or obtained from a static analysis done prior to natural frequency calculations.

Two methods are available in VAST for calculating natural frequencies and mode shapes, namely, the direct iteration method and the subspace iteration method. Generally, the direct iteration method should be used (i) for systems having distinct eigenvalues, and (ii) when only a relatively small number of frequencies are required. Otherwise, the subspace iteration method should be used.

2.4 Dynamic Analysis

PVAST offers the following options for dynamic analysis of structures:

- time history analysis (applied loading or support motion);
- response spectrum analysis;
- frequency response analysis.

2.4.1 Time History Analysis

For PVAST, the governing equations for a time history analysis are:

$$([M]+[M_A]) \{\ddot{\delta}\} + [C] \{\dot{\delta}\} + [K] \{\delta\} = \{f(t)\}$$

where [M] is the structural mass matrix; $[M_A]$ is the added mass matrix (to account for fluid effect); [C] is the damping matrix; [K] is the stiffness matrix; $\{\ddot{\delta}\}$, $\{\dot{\delta}\}$ and $\{\delta\}$ are acceleration, velocity and displacement vectors, respectively; and $\{f(t)\}$ is the time history vector of applied forces (i.e. concentrated or pressure loads). For the case of support motion:

$$\{f(t)\} = -([M] + [M_A])\ \{\ddot{\delta}\}_s$$

where $\{\delta\}_s$ represents support acceleration. In this case $\{\ddot{\delta}\}$, $\{\dot{\delta}\}$ and $\{\delta\}$ represent accelerations, velocities and displacements relative to the support.

Modal Superposition/Direct Integration:

VAST offers two solution methods for a time history analysis; i.e. modal superposition and direct integration. The modal superposition method uses the natural frequencies and mode shapes from an eigenvalue analysis to transform the governing equation into a much simpler form, whereas the direct integration method performs a step-by-step numerical integration of the unmodified equation. The advantage of the modal superposition method lies in the fact that most structural responses can be adequately described by only a few of the lowest natural frequencies. For problems where this is not the case, it may be more economical to use the direct integration method.

Damping for Modal Superposition:

VAST has the capability to include viscous linear damping in the form of a damping ratio (i.e. damping as a ratio to the critical damping) proportional to the velocity vector $\{\dot{\delta}\}$. A value for the damping ratio is to be supplied for each mode used in modal superposition.

Damping for Direct Integration:

VAST offers the capability to include viscous (and linear) damping. A damping matrix defining a linear combination of the mass and stiffness matrices is used, i.e.:

$$[C] = \alpha_M\ [M] + \alpha_K\ [K]$$

where [C] is the damping matrix and α_M and α_K are constants.

If damping is assumed to be proportional to either the mass or stiffness, the non-zero constant (α_M or α_K) can be calculated using one known natural frequency (say ω_m) and the corresponding damping ratio (ζ_m), i.e damping as a ratio to the critical damping:

$$\alpha_K = 0 \; ; \quad \alpha_M = 2\omega_m\zeta_m$$

$$\alpha_M = 0 \; ; \quad \alpha_K = 2\zeta_m/\omega_m$$

For the case when damping is assumed proportional to both mass and stiffness, the constants α_M and α_K can be calculated using two known natural frequencies (say ω_m and ω_n) and the corresponding damping ratios (ζ_m and ζ_n):

$$\alpha_M = 2\omega_m\omega_n\,(\omega_m\zeta_n - \omega_n\zeta_m) \,/\, (\omega_m^2 - \omega_n^2)$$

$$\alpha_K = 2(\omega_m\zeta_m - \omega_n\zeta_n) \,/\, (\omega_m^2\ \omega_n^2)$$

Inherent in this formulation of damping is the following expression for damping ratio:

$$\zeta_i = 1/2(\alpha_M/\omega_i + \alpha_K\omega_i)$$

which can be used to obtain the damping ratio for all known natural frequencies based on knowledge of the damping coefficients in one or two modes (i.e. the damping ratio necessary to calculate α_M and α_K).

2.4.2 Response Spectrum Analysis

PVAST offers the capability to perform a response spectrum analysis using previously calculated natural frequencies and mode shapes. Either a displacement vs frequency or acceleration vs frequency shock spectrum can be input. The modal responses are combined using one of three methods, i.e. ABS (absolute sum); RMS (root mean square); and PRMS (largest modal contribution plus the root mean square of the rest).

2.4.3 Frequency Response Analysis

The frequency response analysis option for PVAST provides solution of the steady state displacements of a structure subjected to a set of sinusoidal loads of known amplitudes and frequencies, or alternately to a more general periodic loading of known Fourier Series representation, using previously calculated natural frequencies and mode shapes. (At the present time PVAST is limited to the odd terms in Fourier Series representation.)

Loads are specified as amplitudes and can consist of concentrated forces and distributed pressures.

2.5 Substructures

The combined blade and root models represent a large problem which, when using standard solution methods, require large computer storage and execution times. One way of overcoming this difficulty is to partition the structure into a number of smaller substructures, and then link them together to obtain the complete solution.

For PVAST, substructuring is handled by what is termed the substructure-/superelement technique. A substructure is defined as a finite element model of a structural unit having its own independent node numbering scheme. One substructure can be used to represent one or more identical structural units located within the overall structure. A superelement is created from a substructure by selecting a set of master nodes from the regular nodes and eliminating all degrees-of-freedom not associated with the master nodes. Master nodes consist of regular nodes at boundaries common to other superelements, and regular nodes which are to have lumped masses and/or prescribed displacements. Also, for natural frequency and dynamic response analysis, master nodes will include regular nodes associated with dynamic degrees-of-freedom necessary to characterize the deformation of the superelement in the modes of interest.

Superelements are treated in the same way as regular finite elements. Superelement matrices (which are formed by reducing the assembled stiffness and mass matrices of the substructures) are assembled to form the equilibrium equations for the entire structure. These equations relate to the degrees-of-freedom associated with the master nodes and represent a substantial reduction in size as compared to modelling the entire structure without substructuring.

2.6 Numerical Error Estimation

Among the most reliable methods of estimating accuracy loss incurred during solution of a simultaneous linear equilibrium equation system involves the condition number of its coefficient matrix. The condition number is simply the ratio of the largest and smallest eigenvalues of the coefficient matrix and taking its logarithm yields an upper bound on the number of significant figures lost. A capability is provided within VAST to compute condition numbers of structural stiffness matrices and thereby to estimate solution errors for the equilibrium equation systems associated with static analysis. (At present, VAST does not offer a capability to estimate errors associated with eigenvalue analysis and dynamic analysis.)

When the upper bound on the number of significant figures lost approaches the number of digits to which numbers are stored on the computer, a high degree of unreliability in the results obtained is indicated. In this event, it is recommended that the finite element model be carefully studied with a view to identifying and eliminating those modelling characteristics responsible for serious accuracy deterioration.

3.0 FINITE ELEMENT LIBRARY

3.1 Structural Elements

The library of structural elements for the analysis of propellers consists of six different element types. They are:

- A thick/thin shell element for modelling propeller blades;
- An isoparametric three-dimensional 20-noded element including various degenerate forms such as wedges for modelling propeller blades and/or roots (i.e. palms and hubs);
- An isoparametric 10-noded tetrahedron element to be used in fillet modelling;
- A transition element to connect the thick/thin shell and 20-noded solid elements;
- A triangular membrane element for solving two-dimensional stress problems (eg. keyway model);
- A rod element to model bolts, including the bolt tightening loads;
- A gap element to transmit only compressive forces (eg. palm support boundaries).

3.1.1 Thick/Thin Shell Element

The shell element is the one introduced in 1968 by Ahmad and co-workers [3]. The element is based on an isoparametric formulation and has proven to be extremely useful in curved thick shell problems, and in thin shell problems when a reduced numerical integration technique is employed during the formulation of the element stiffness [4]. The element avoids the common assumption that normals remain normal after deformation, thus permitting the shell to experience shear deformation, an important feature in thick shell situations. The detailed formulation of the element is given in References [3] and [4].

Two basic co-ordinate systems are used in the derivation of the shell element, i.e. the global (reference) Cartesian co-ordinate system XYZ, and a natural co-ordinate system $\xi \eta \zeta$ (see Figure 3.1). The geometric model for the element is shown in Figure 3.1(a). Known XYZ co-ordinates for eight upper surface nodes and eight lower surface nodes, together with a quadratic interpolation function in the surface co-ordinates ($\xi\eta$) and a linear interpolation function through the thickness (ζ) are used to describe the element's geometry.

The displacement model for the element is shown in Figure 3.1(b). The overall displacement of the element is described in terms of five degrees-of-freedom at eight mid-surface nodes, i.e. three translations, u, v, and w, and two rotations, α and β, with the additional assumption that the strain in the thickness direction is negligible. Displacements throughout the element are described using a quadratic interpolation function in the surface co-ordinates ($\xi\eta$).

It is necessary to define more clearly the rotational degrees-of-freedom, α and β. Actually, they are the rotations of $\overline{V}_3$ - a vector joining a nodal pair about two orthogonal directions normal to it. An infinite number of possibilities exist for generating these two rotations. For the present element, the direction $\overline{V}_1$, corresponding to β, is defined as being normal to a plane of reference defined by the Vector $\overline{V}_3$ and the global X axis. The direction $\overline{V}_2$, corresponding to β, is defined as being normal to the plane defined by the vectors $\overline{V}_3$ and $\overline{V}_1$. The situation is more clearly defined in Figure 3.2(a). Where $\overline{V}_3$ is parallel to the global X axis, $\overline{V}_1$ is defined as being normal to a plane defined by $\overline{V}_3$ and the global Y axis $\overline{V}_2$ normal to the plane formed by $\overline{V}_3$and $\overline{V}_1$.

The geometry of the element is defined at eight corner nodes and eight mid-side nodes, for a total of 16 nodes. The displacements are defined at eight mid-plane nodes; i.e. four corner nodes and four mid-side nodes. With regard to the location of mid-side nodes, a node between two corner nodes, say i and j, separated by a distance L, must be positioned with respect to node i, a distance greater than 0.25L, but less than 0.75L. In the present formulation the mid-side nodes are positioned at approximately 0.5L with respect to either corner node.

It is essential that the nodes of the element be numbered in a conventional manner. Referring to Figure 3.1, the first two numbered corner nodes define the direction of the axis ξ, whereas the second and third numbered corner nodes define the direction of the axis η. The axis ζ completes the right-handed co-ordinate system. The upper surface of the element corresponds to $\zeta = +1$, and the lower surface corresponds to $\zeta = -1$. It will be observed that, when looking at the upper surface of the geometric model, Figure 3.1(a), both this surface and the lower surface are numbered counterclockwise, the corner nodes being numbered first, then the mid-side nodes last. The same is true for the mid-plane of the displacement model, Figure 3.1(b). Also, it will be noticed that for the case of the geometric model, all the upper surface nodes are numbered first. This is essential and, in fact, applies to an assemblage of the elements, in which case the entire upper surface must be numbered first.

The numerical integration required in the formulation of the stiffness matrix and consistent mass matrix and load vector for the element is performed using the minimum number of integration points necessary for convergence, which for the case of the shell element is eight [4]; i.e. four on each side of the mid-plane of the element.

Consistent loads due to body forces arising from centrifugal and gravitational forces are computed. Also, loads due to distributed external pressures are computed using the geometry interpolation functions. Hence, a quadratic variation in pressure loading can be represented. The pressure is specified at the upper and lower surface nodes and is defined as being positive towards the element interior.

Element stresses are determined with respect to a local Cartesian co-ordinate system x'y'z'. This local co-ordinate system coincides with the vectors $\overline{V}_1$, $\overline{V}_2$, $\overline{V}_3$, as shown in Figure 3.2(b). Hence, the x' and y' axes are tangent to any ζ = constant surface. The stresses consist of the two normal components $\sigma_{x'x'}$ and $\sigma_{y'y'}$, and the three shear components $\sigma_{x'y'}$, $\sigma_{x'z'}$ and $\sigma_{y'z'}$ and are computed at the numerical integration points. These stress components are then interpolated to corner nodes on the upper and lower surfaces using the stress smoothing technique described in Reference [5]. These stresses, along with their corresponding principal stresses σ_1 and σ_2 (and their directions, defined by the angle between the x' axis and the direction of σ_1, which is the primary principal stress) are provided in the computer printout.

3.1.2 Three-Dimensional Solid Element

The isoparametric three-dimensional solid element is shown in Figure 3.3. The theoretical formulation for the element is given in Reference [5]. The element has a total of 20 nodes, and material properties are assumed to be isotropic. Two forms of the element are available, i.e. a brick element and a wedge element.

Six stress components relative to the global co-ordinate system (σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz}, σ_{xz}) are computed at the numerical integration points. These stress components are then interpolated to the eight corner nodes using the stress smoothing technique described in Reference [5]. These stresses, along with the corresponding principal stresses σ_1, σ_2, and σ_3 are provided in the computer printout.

3.1.3 Tetrahedron Element

The tetrahedron element, shown in Figure 3.4, is provided as a companion element for the three dimensional wedge element. Its theoretical formulation is similar to that for other isoparametric elements already considered with the exception that volume coordinates L_1, L_2, L_3, and L_4 define the local coordinate system (see Reference [11]). Typically the local coordinate L_i is 1 at node i and decreases linearly to 0 at the remaining vertices and the face sharing those vertices. The default integration formula used to evaluate element matrices involves five element points expressed of course relative to the volume coordinate system - the first four are located symmetrically with respect to volume coordinates, whereas the fifth is at the element centroid.

Six stress components relative to the global coordinate system are computed at the first four numerical integration points. Stress smoothing similar to that for the thick/thin shell element and the three dimensional solid element has also been implemented. In particular, linear shape functions with respect to the local coordinates extrapolate stresses from four integration points to the element vertices. These stresses as well as the corresponding principal stresses are printed.

3.1.4 Transition Element

The transition element is shown in Figure 3.5. The element is used to connect the thick/thin shell element to the three-dimensional solid element when the shell is used to model the blade. The element has one edge surface with four corner nodes and four mid-side nodes, which is compatible with the three-dimensional solid. The opposite edge is compatible with the thick/thin shell element. The other two opposite edges are compatible for connecting a row of transition elements. Element stress output is the same as the three-dimensional solid element. The detailed formulation of the element is given in Reference [8].

3.1.5 Triangular Membrane Element

The triangular membrane is a 3-noded constant strain element and is similar to the one derived in Reference [9] (Chapter 5, Section 5.7). The element is shown in Figure 3.6.

3.1.6 Rod Element

The rod element is a 2 noded-axial load carrying member with initial strain carrying capability. Its derivation can be found in Reference [9] (Chapter 5, Section 5.2). The element is shown in Figure 3.7.

3.1.7 Gap Element

The gap element is used to specify that contact be made between two degrees-of--freedom. The element is defined by: (i) two nodes, N1 and N2, which specify two points that may come into contact during loading; (ii) a degree-of-freedom, i, which defines the direction of resistance once contact is made; and (iii) the gap width, G. In the present formulation, the direction of contact must be along either the global X, Y or Z axes, with i equal to 1, 2 or 3, respectively. N1 and N2 must be numbered so that the direction from N1 to N2 is in the positive X, Y or Z direction. Shown in Figure 3-8 below is a typical gap element used to specify contact in the direction of the Y axis. The derivation for the element is given in Reference [8]. Currently, the gap element can only be used for static analysis.

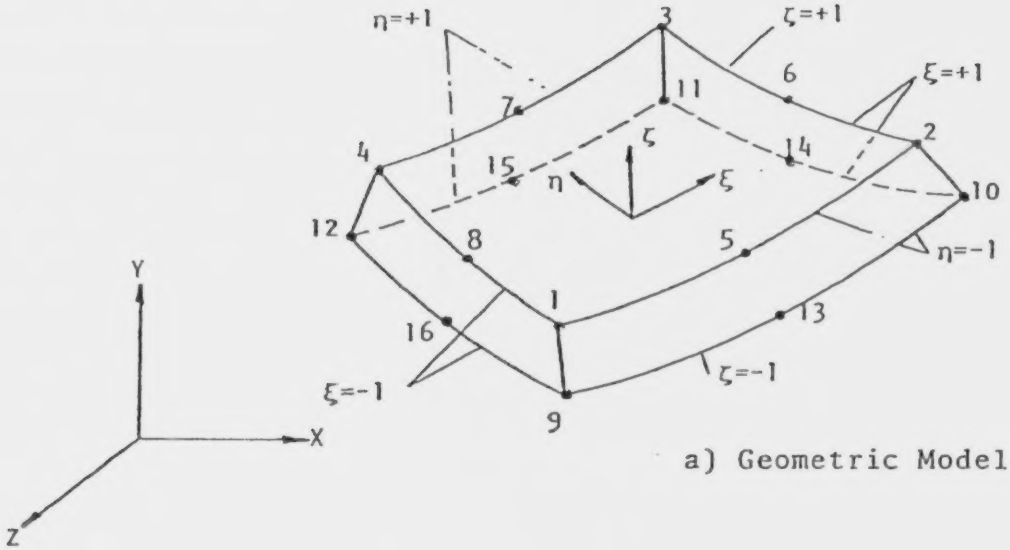
3.2 Fluid Elements

The program offers four elements for calculating the added mass of submerged structures. These elements are shown in Figure 3.9.

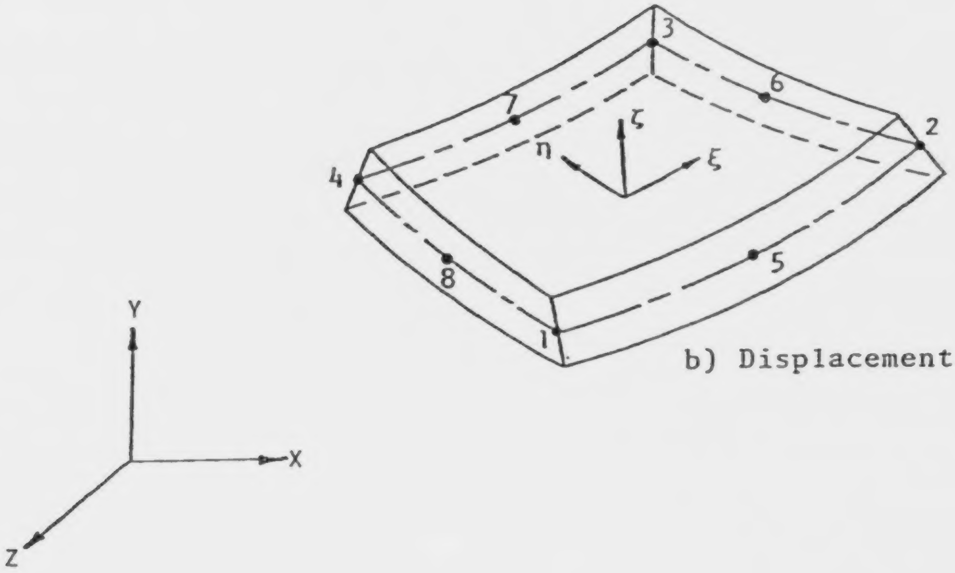
Two types of fluid elements are available for modelling the surrounding fluid, i.e. a simple 8-noded element and a more complex 20-noded element. These elements are shown in Figure 3.9(a). Two types of interface elements are available and are shown in Figure 3.9(b). The 4-noded interface element is used in conjunction with the 8-noded, fluid element, while the 8-noded interface element is used in conjunction with the 20-noded fluid element.

The node numbering schemes required for these elements are defined in Figure 3.9. Details on element derivation are given in Reference [2].

PVAST also provides the capability to model the added mass effects by surface panels. The panel node numbering scheme is shown in Figure 3.10. The proper node numbering scheme for the panel is that when viewing the panel from the fluid side, the nodes are entered counterclockwise [7]. Details of the surface panel formulation are given in Reference [12].

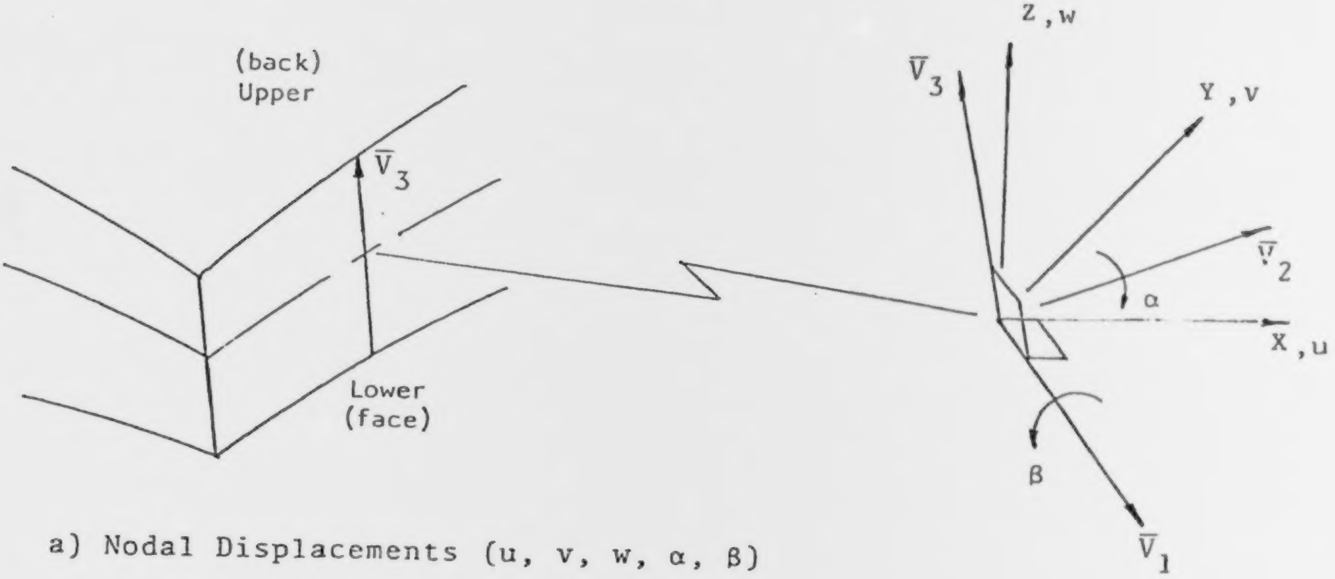

η=+1
3
ζ=+1
6
ξ=+1
7
11
2
4
15
ζ
η
ξ
14
10
12
8
5
η=−1
Y
1
13
16
ξ=−1
ζ=−1
9
X
Z

a) Geometric Model

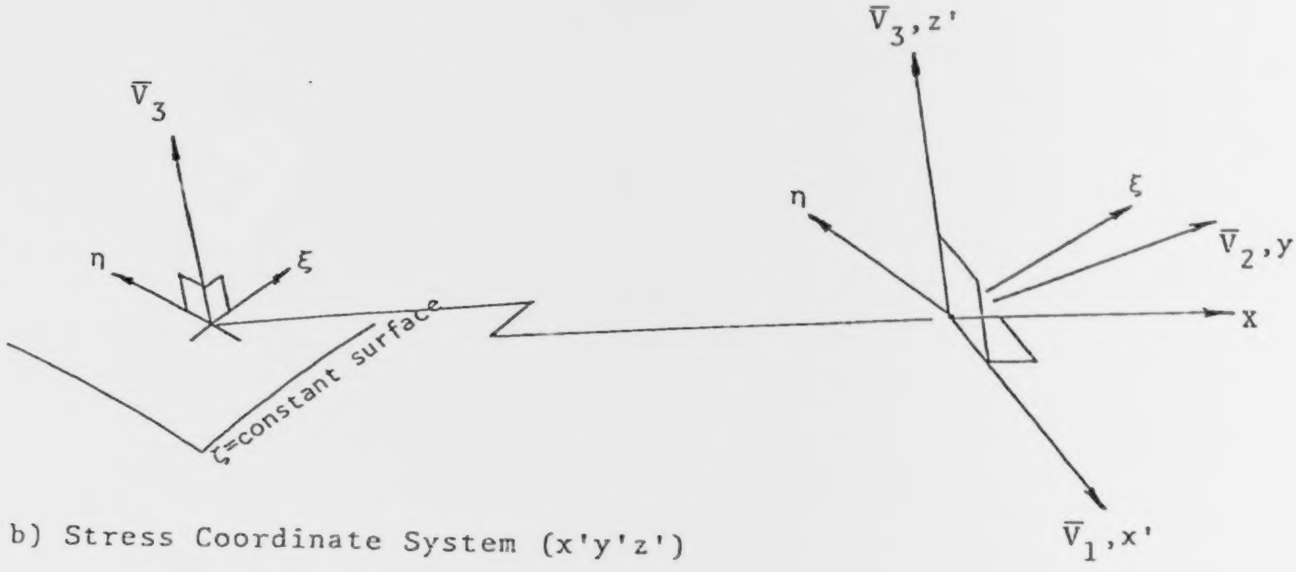

3
6
7
ζ
2
η
ξ
4
8
5
1
Y
X
Z

b) Displacement Model

FIGURE 3.1: Thick/Thin Shell Element

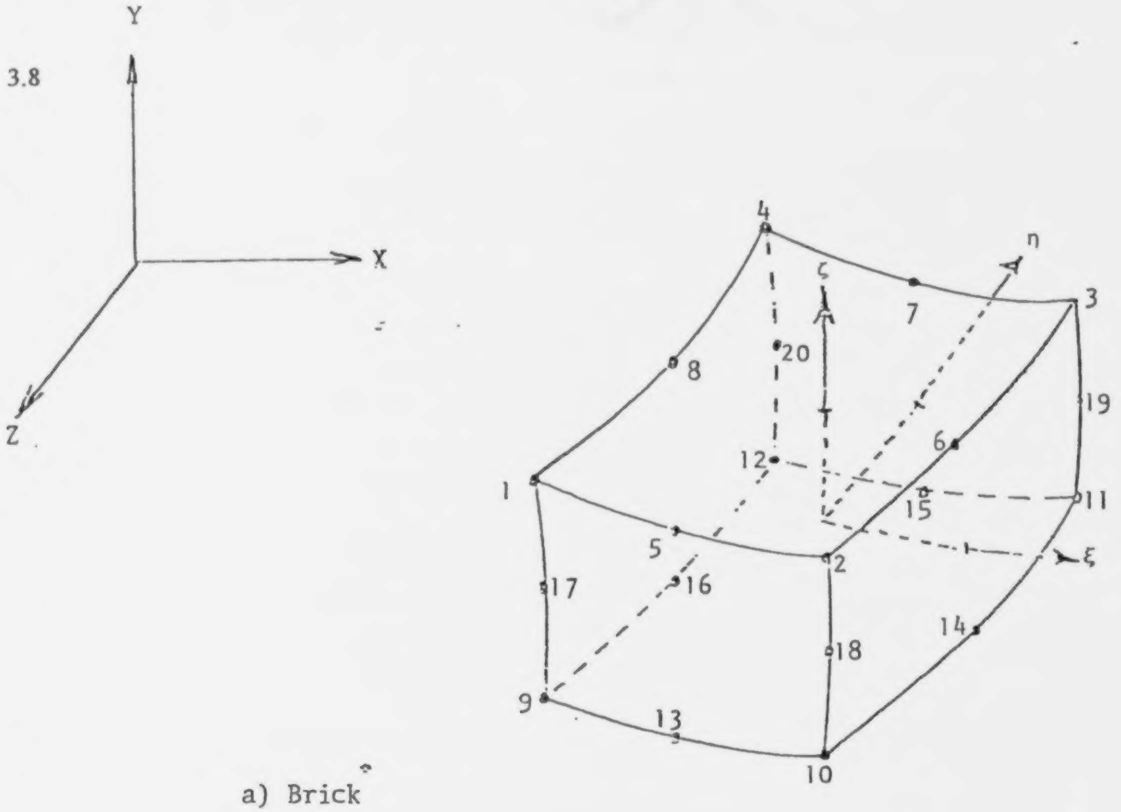

(back)
Upper
$\bar{V}_3$
Lower
(face)
Z , w
$\bar{V}_3$
Y , v
$\bar{V}_2$
α
X , u
β
$\bar{V}_1$

a) Nodal Displacements (u, v, w, α, β)

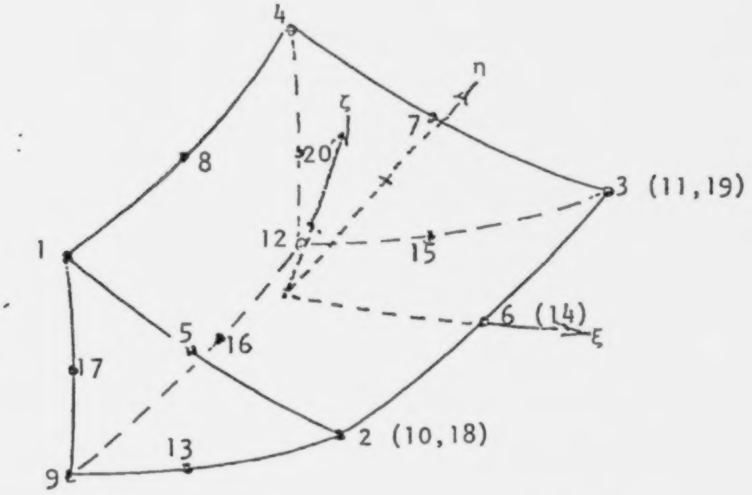

$\bar{V}_3$
η
ξ
ζ=constant surface
$\bar{V}_3$, z'
η
ξ
$\bar{V}_2$, y'
X
$\bar{V}_1$, x'

b) Stress Coordinate System (x'y'z')

FIGURE 3.2: Coordinate System for Shell Element


Y
X
Z
4
η
ζ
7
3
20
8
19
12
6
1
15
11
5
2
ξ
17
16
14
18
9
13
10

a) Brick


4
η
ζ
7
20
8
3 (11,19)
1
12
15
6 (14)
ξ
5
16
17
2 (10,18)
13
9

b) Wedge

FIGURE 3.3: Isoparametric 3-D Element

$$L_1 = \frac{\text{Volume P234}}{\text{Volume 1234}}$$

$$L_2 = \frac{\text{Volume 1P34}}{\text{Volume 1234}}$$

$$L_3 = \frac{\text{Volume 12P4}}{\text{Volume 1234}}$$

$$L_4 = \frac{\text{Volume 123P}}{\text{Volume 1234}}$$

$$L_1 + L_2 + L_3 + L_4 = 1$$

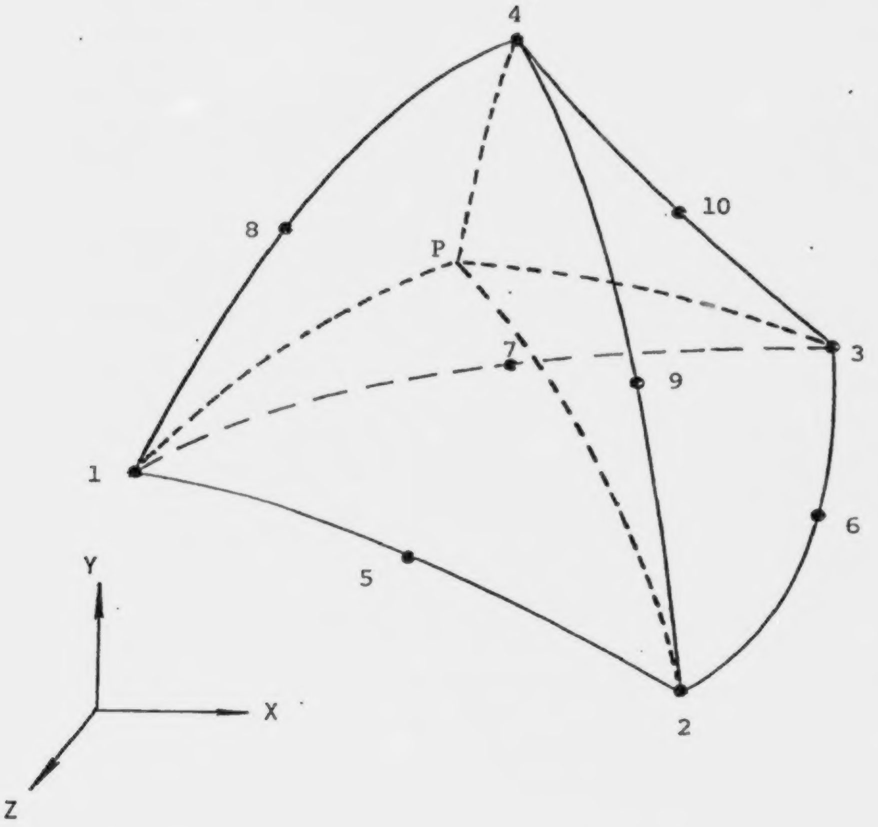
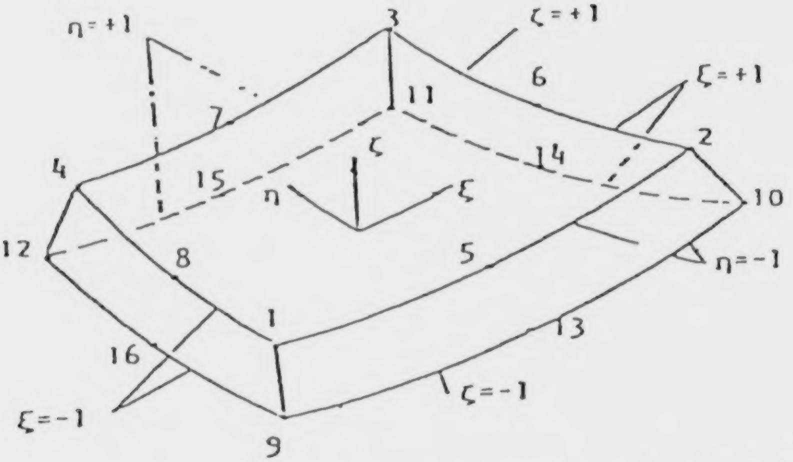
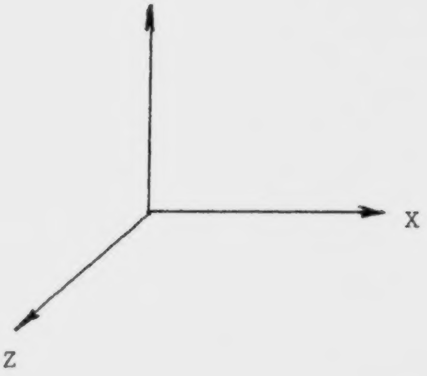
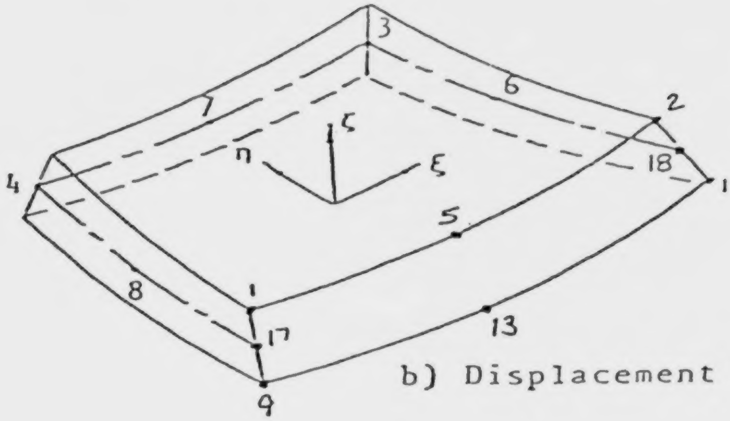


FIGURE 3.4: Tetrahedron Element


η = +1
3
ζ = +1
6
ξ = +1
11
7
2
4
ζ
14
15
η
ξ
10
12
8
5
η = -1
1
13
16
ζ = -1
ξ = -1
9

a) Geometric Model




3
6
2
7
ζ
η
ξ
18
4
10
5
8
1
17
13
9

b) Displacement Model

FIGURE 3.5: Transition Element

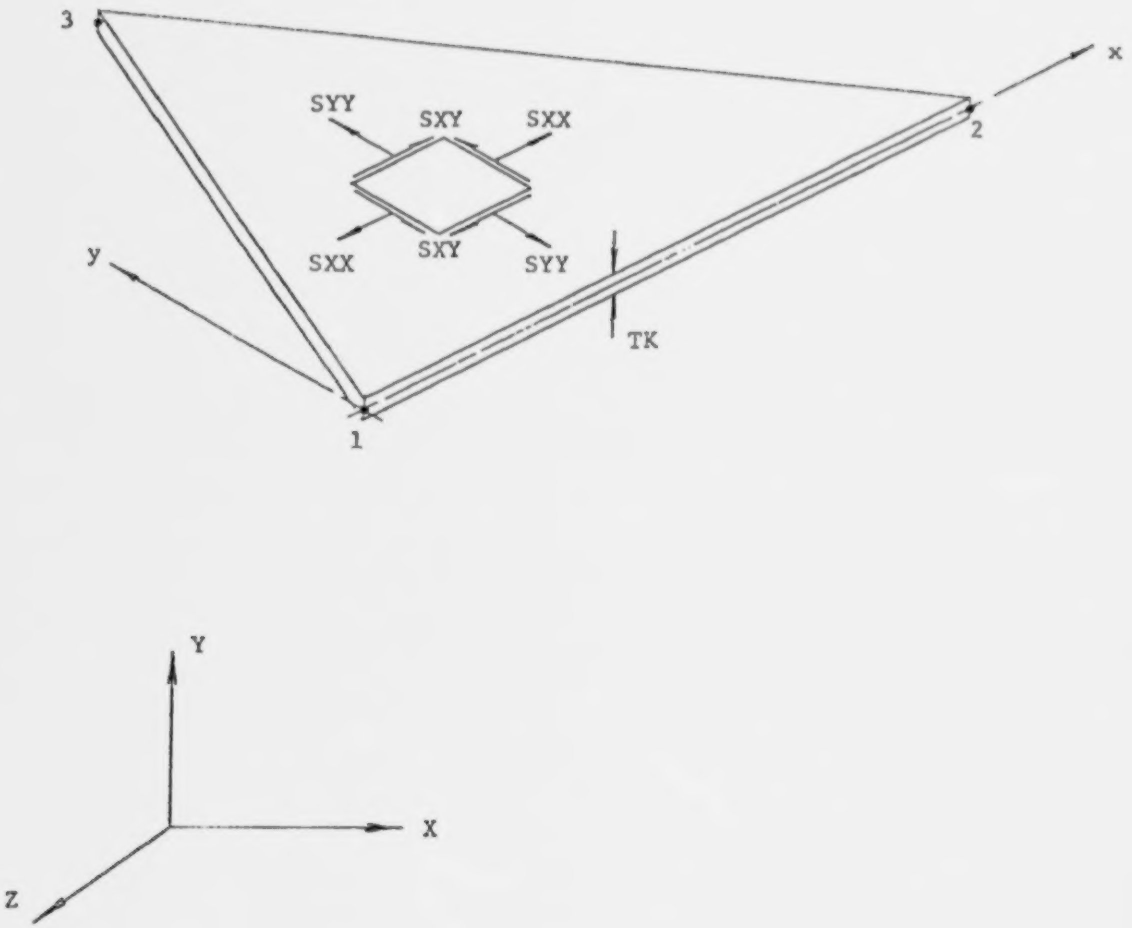

3
x
SYY
SXY
SXX
2
y
SXX
SXY
SYY
TK
1
Y
X
Z

FIGURE 3.6: Triangular Membrane Element

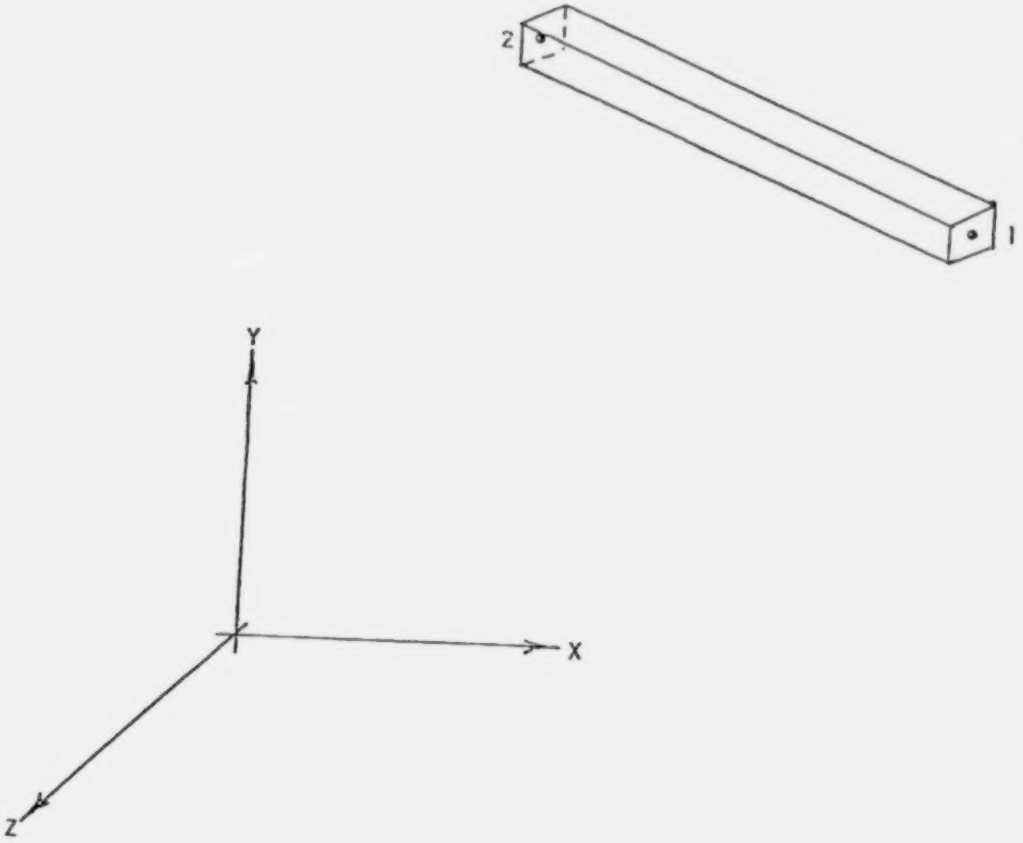


FIGURE 3.7: Bar Element

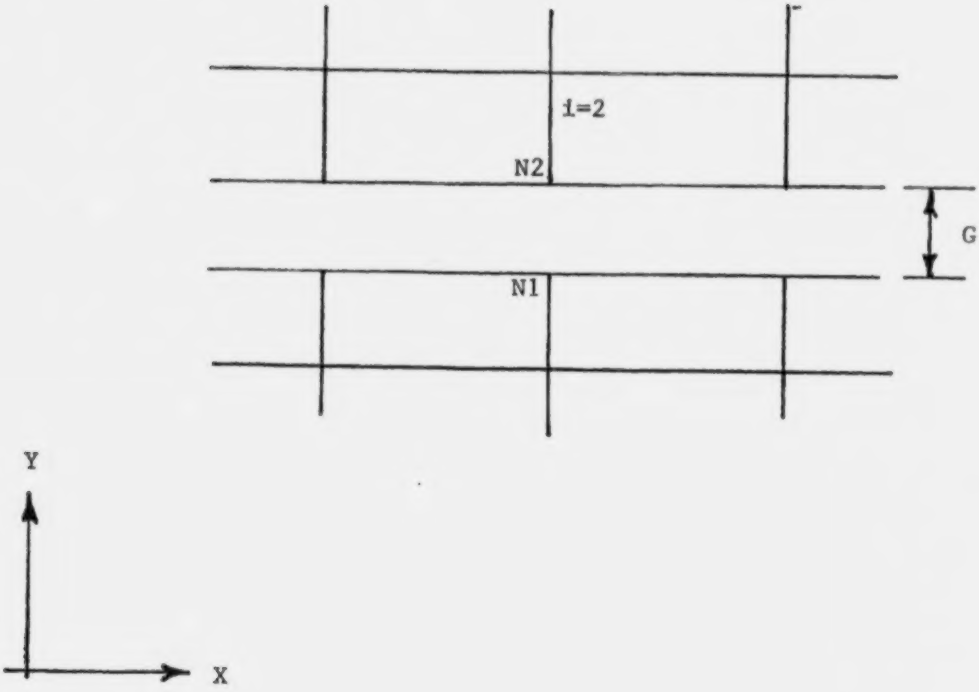
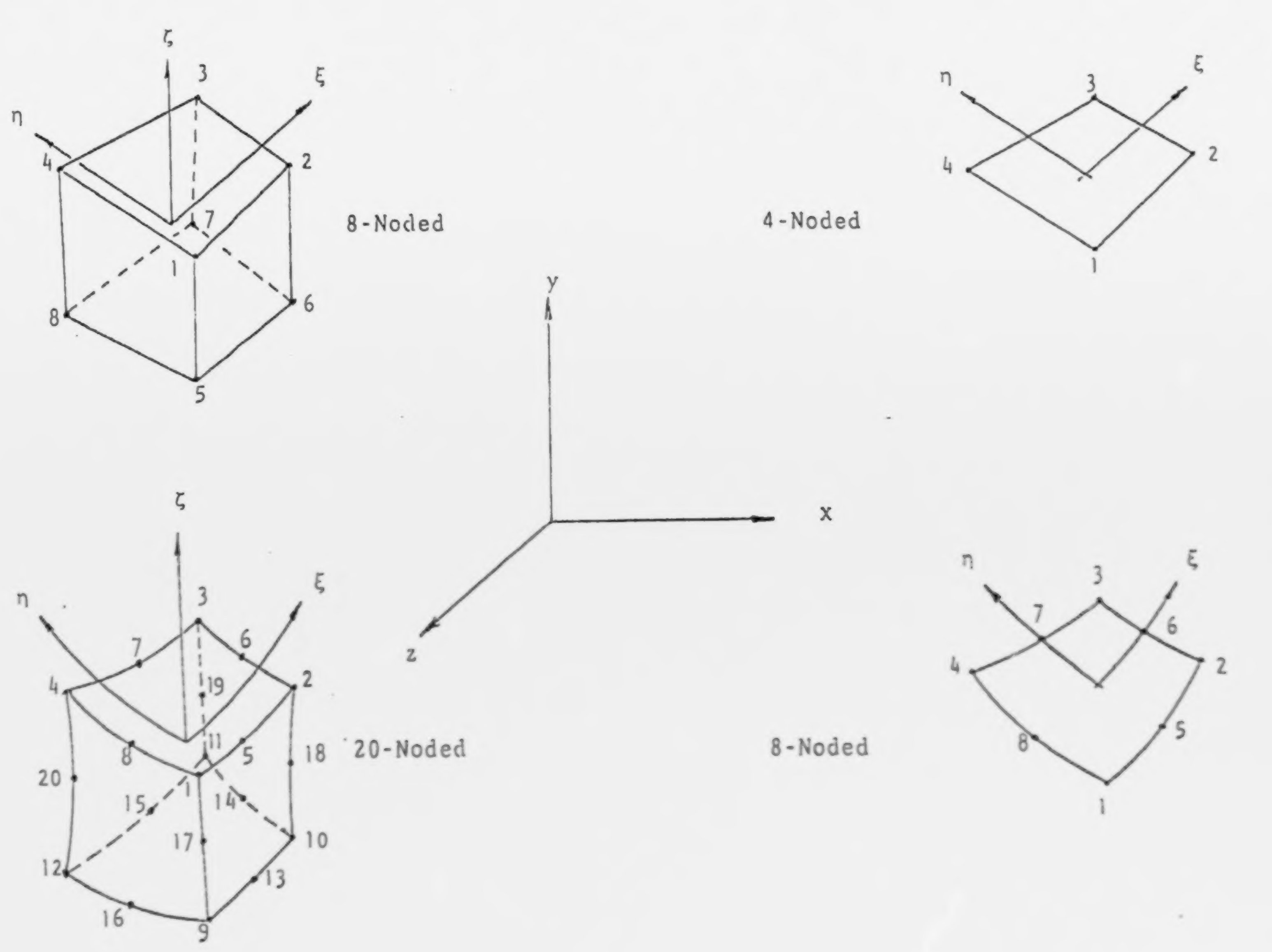


FIGURE 3.8: Gap Element


ζ
η
ξ
1
2
3
4
5
6
7
8
8-Noded
4-Noded
y
x
z
9
10
11
12
13
14
15
16
17
18
19
20
20-Noded
8-Noded

a) Fluid Elements

b) Interface Elements

FIGURE 3.9: Fluid Finite Elements

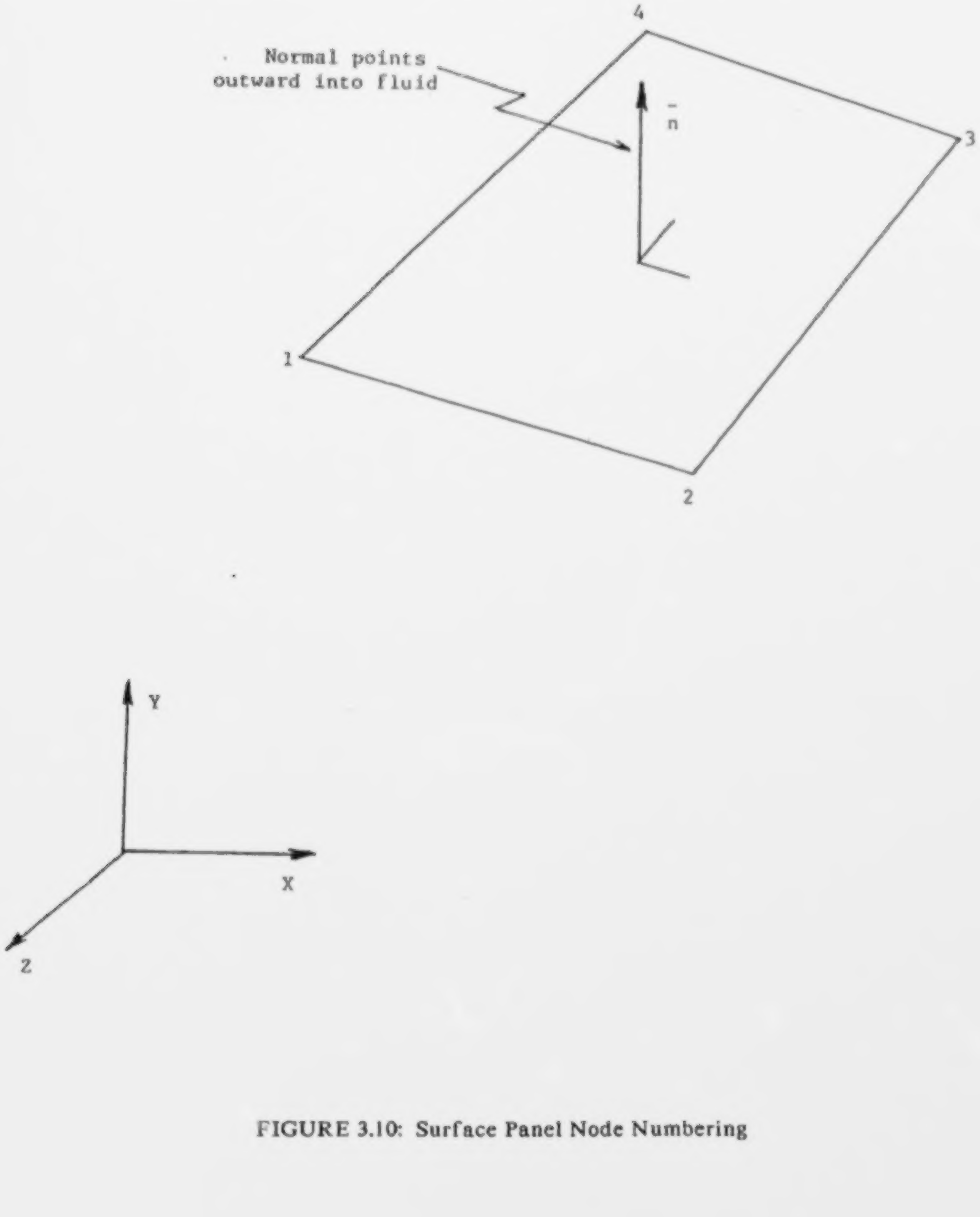

4
Normal points
outward into fluid
$\bar{n}$
3
1
2
Y
X
Z

FIGURE 3.10: Surface Panel Node Numbering

4.0 PROPELLER MODELLING

4.1 Introduction

The types of analysis which can be performed by PVAST range from a relatively simple single blade analysis to a very complex analysis of a complete blade-hub assembly. The more sophisticated an analysis is, however, the more costly it will be. The user should keep this in mind when selecting the various modelling options available. Suggested guidelines relating to model selection are provided in Section 8.

A primary concern in any finite element analysis is accuracy. Generally, the accuracy of the finite element method increases as element sizes become smaller (i.e. meshes become finer). However, finer meshes mean more elements, more degrees-of-freedom, increased matrix sizes, and, consequently, larger costs. In PVAST, the fineness of the mesh is defined and controlled by the user. Again, guidelines are provided in Section 8.

Accuracy does not depend on element size alone. Errors related to structural idealization and data manipulation are closely associated with accuracy. Manipulation errors are those associated with mathematical operations, such as round-off and truncation. Idealization errors are those associated with grid and element selection (eg. the use of shell elements to represent thick regions, which actually should be represented by solid elements). Manipulation errors are discussed more fully in Reference [6].

4.2 Structural Models

4.2.1 Geometry Considerations

Propellers can be treated as consisting of three basic parts: (i) the blade section, (ii) the hub, and (iii) the part which provides the means of attaching the blade section to the hub. In this manual, the third part is referred to as the root. For monobloc (fixed pitch) propellers, the root is the transition region (fillet area) between the blade section and the hub. For controllable pitch propellers (detachable blades), the root consists of a flange or palm, which is bolted to the hub, and a transition region (fillet area) between the blade section and the palm. Some controllable pitch propellers have overhang at the root. A typical controllable pitch propeller with overhang is shown in Figure 4.1.

4.2.2 Modelling Options

PVAST offers a wide variety of modelling options, ranging from the modelling of the propeller blade alone, to the modelling of the combined blade, palm, bolts and boltholes of a controllable pitch propeller or the combined blades and hub of a fixed pitch propeller.

The blade can be modelled with either the shell or the 3-D solid element. The shell element is especially useful for thin blades and for special blades such as those used for supercavitating propellers, which have a very thin leading edge and a thick trailing edge. When the shell element is used a special transitional element is employed to make the transition from the shell to the 3-D solid element used to model the fillet and the palm or hub. Tetrahedron and wedge elements are also used in the fillet and palm/hub regions. Gap, rod and plane stress/strain elements are used to solve special modelling problems (eg. blade carrier interface and hub keyway).

A typical blade model is shown in Figure 4.2(a). When interest is confined to only blade stresses and/or natural frequencies, it will usually be sufficient to use models of this type. Even for a relatively fine grid, the computer costs will not be excessive. An additional feature is available which offers a more detailed blade model by incorporating a portion of the root fillet to form an additional row of elements in the chordwise direction (see Figure 4.2(b)). This model offers a means for taking into account root flexibility. However, stresses computed in this area should be interpreted with caution, since only one element is used through the thickness.

If a palm or hub is to be included in the model, computer costs will be large compared to a blade only analysis. For combined blade-palm or blade- hub analysis, PVAST generates a substructure for the blade and a substructure for the palm or a segment of the hub. Typical models are shown in Figures 4.3 and 4.4. In the case of a fixed pitch propeller, the blade and hub segment substructures can be assembled to model the entire propeller, as shown in Figure 4.5.

4.2.3 Attachment Bolts

It has become important with the introduction of the variable pitch propeller to determine attachment bolt loads and stresses. With the use of conventional methods for restraining boundary nodes the correct stress distribution at the blade carrier interface cannot be determined. PVAST has been provided with pseudo-gap elements for static analysis to overcome this contact problem. By the use of gap elements at the bolt head-palm inter faces and between the blade-carrier interface, it is possible to identify when contact is made between interfaces and establish that the resulting forces are compression. If separation occurs, the forces at the interface are redistributed because the gap element cannot transmit tension forces. Static equilibrium is maintained by distributing the load among the bolts, which can carry only tension and shear.

In PVAST, two methods are available for incorporating bolts. One is a relatively simple approach whereby the bolts are represented by rod elements. In order to use this method, a finite element model of the palm must be generated. Prestresses can be incorporated to account for the bolt tightening loads. Gap elements are used to model the palm/hub interface. The other method is a much more sophisticated approach involving detailed substructures of bolts and bolthole regions, see Figure 4.6. The latter method is very costly, with regard to computer time, and should be used only if a detailed stress analysis of the bolt and stress concentrations in the bolthole region are required.

4.2.4 Hub Keyway Model

A hub keyway can often be a region of high stress due to the localized reduction in hub section and the stress concentrations present. Strictly speaking, a keyway represents a complex three-dimensional problem. However, to treat it as such, would be, if not impossible, extremely complicated and not very practical. Consequently, PVAST provides the capability to perform a plane stress/strain analysis of keyways. A pre-processor program generates the model shown in Figure 4.7. The grid is generated with a much higher density in the region surrounding the keyway to account for high stress concentrations. The user provides the inner and outer hub radii, the hub length, the depth and width of the keyway and blade loading.

4.2.5 Air Channel Model

Some propeller blades are provided with an air channel near the blade leading edge, as shown in Figure 4.8(a). In PVAST, an option has been provided to model this air bleed channel. The approach adopted here is that the blade is first modelled and analyzed ignoring the channel effects and then a detailed model of a portion of the blade containing the air channel is generated as shown in Figure 4.8(b). The user provides the location of the channel model as well as the dimensions necessary to generate it. Since the channel model is a 3-dimensional structure it can only be generated from a blade that has been modelled by the 20-noded solid elements. The user has some control on the meshing of the air channel model.

4.3 Modelling Surrounding Fluid

4.3.1 Fluid Elements

Stated simply, the modelling of water surrounding the propeller involves the idealization of a finite volume of the water using the 8-noded or 20-noded fluid elements described in Section 3.2. The boundaries of this volume of water must be far enough away from the blade so that the boundary condition of zero velocity potential is approximately satisfied. The finite element mesh, of course, must be fine enough to give sufficiently-accurate results; but, here again, cost may be the overriding factor, dictating just how fine a mesh can be made.

A view of a fluid finite element mesh for a typical propeller blade which seems to give satisfactory results (Reference [2]) is shown in Figure 4.9. The solid lines in Figure 4.9 outline the structural finite element grid; the dotted lines outline the fluid grid. The complete model consists of several layers of fluid elements on each of the two faces of the blade (see Figure 4.10).

The model can be constructed using either the 8-noded fluid element or the 20-noded fluid element. Although the 8-noded element has fewer degrees-of-freedom than the 20-noded element, more layers of the 8-noded elements would be required to obtain comparable accuracy. The 20-noded element has the added advantage that it is compatible (in terms of the number of face nodes) with the shell element used to model the blade itself. Therefore, it is recommended that the 20-noded fluid element be used to model the fluid unless, of course, only a crude approximation of the added mass is needed. Then the 8-noded fluid element should be used.

In order to approximately satisfy the boundary condition of zero velocity potential, a depth for the fluid model equal to a typical dimension of the blade should be used. Also, for most cases, a maximum of six layers of elements (i.e. three on each face of the blade) is recommended.

On the surface of the propeller blade, interface elements are defined. When specifying the nodes of these elements, two numbers for each node must be given, i.e. the number of the node in the fluid model and the number of the node (displacement) in the structural model. Also, interface elements for both the face and back surfaces must be provided. Consequently, the number of interface elements should be twice the number of blade (shell or 20-noded) elements.

4.3.2 Surface Panel Method

The effort, storage and CPU time required in using fluid finite elements to generate the fluid added mass can often be prohibitive. For this reason PVAST provides an option to compute the added mass effect using the surface panel method. This is a boundary integral approach in which the velocity potential in the fluid surrounding the structure is represented in the form of sources or dipoles on the fluid-structure interface. The panels consist of four corner nodes and are provided at both the face and back surfaces of the blade, so that the number of surface panels is equal to twice the number of blade elements.

Although the surface panel method requires less computer storage and CPU time, it should be used with some caution as it may produce poor results in some situations where the fluid finite element method would otherwise provide greater accuracy [12].

1. Propeller blade
2. Blade bolt
3. Fillet area (root)
4. Propeller flange or palm
5. Hub
6. Blade overhang

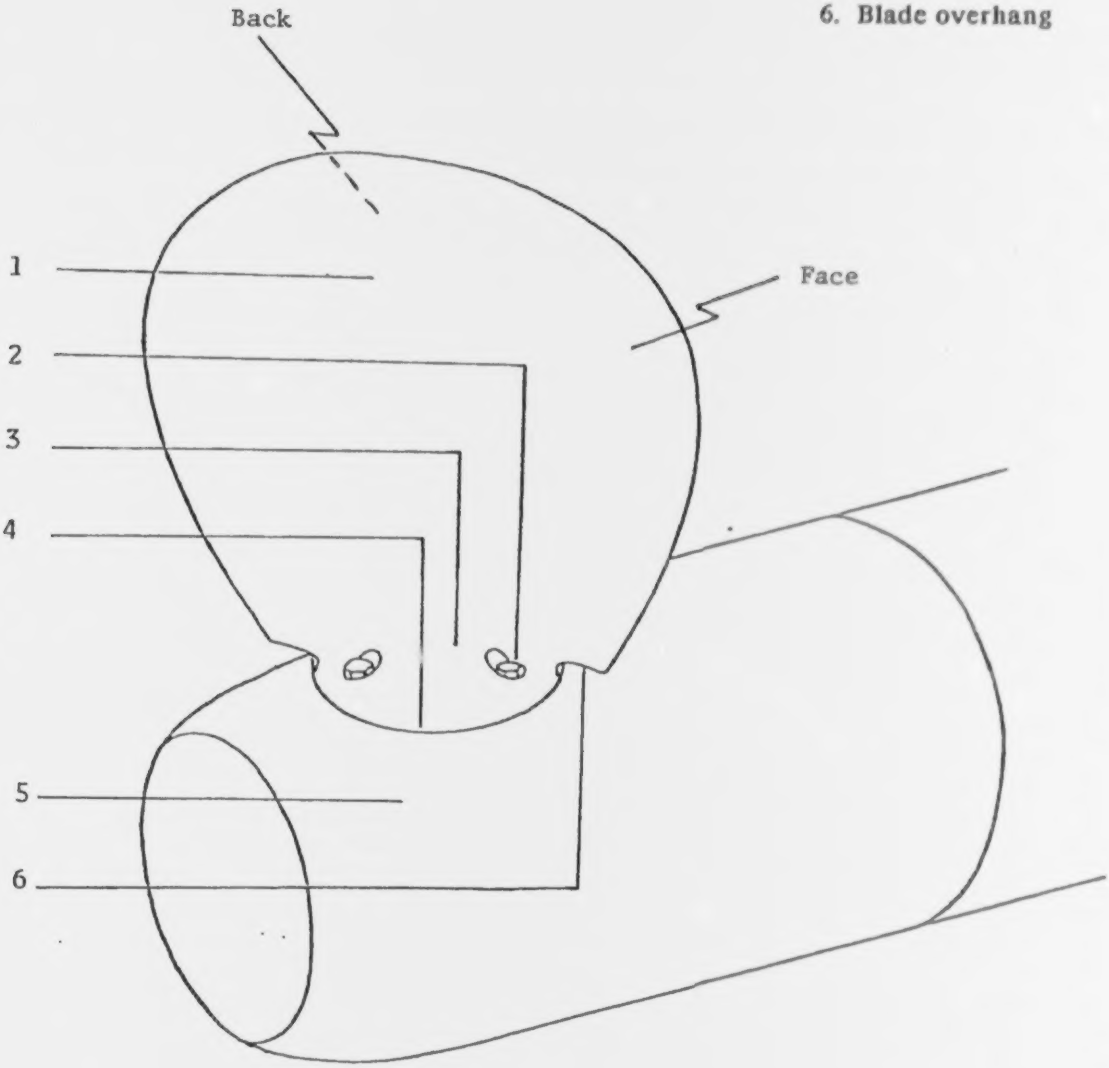
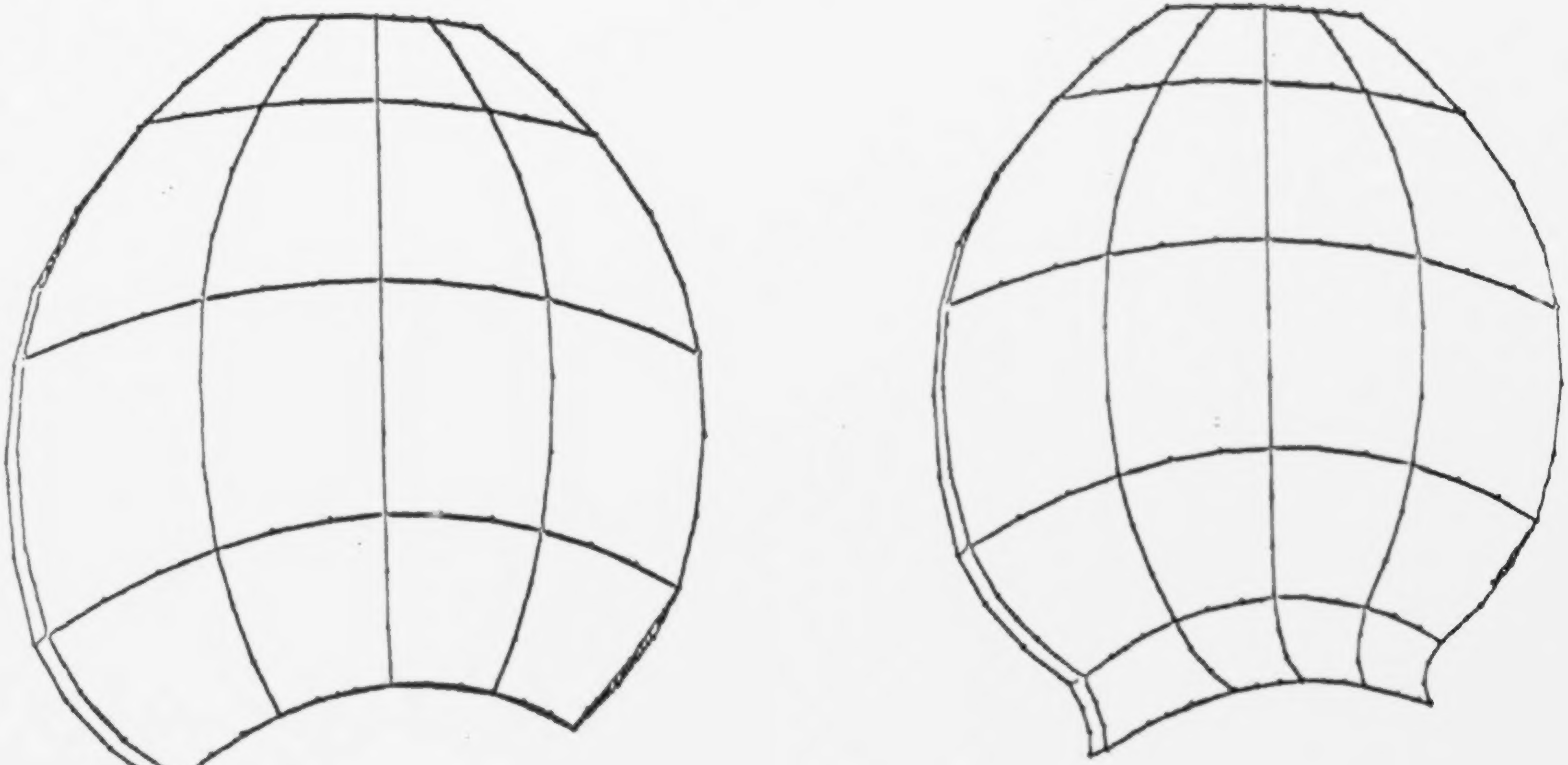


FIGURE 4.1: Typical Propeller with Overhang



a) Without Fillet

b) With Portion of Fillet

FIGURE 4.2: Blade Models

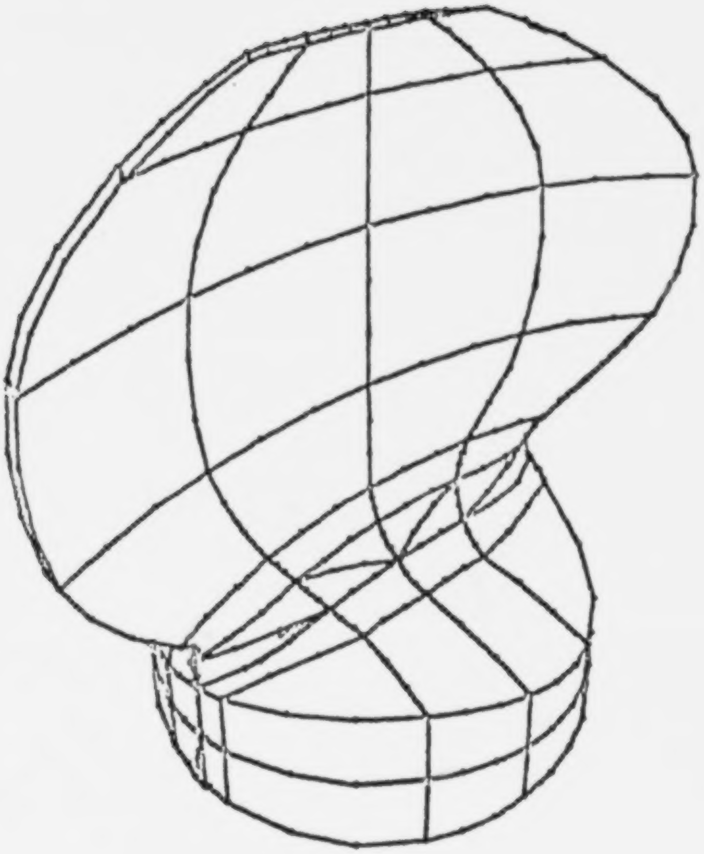


FIGURE 4.3: Substructured Blade and Palm Model

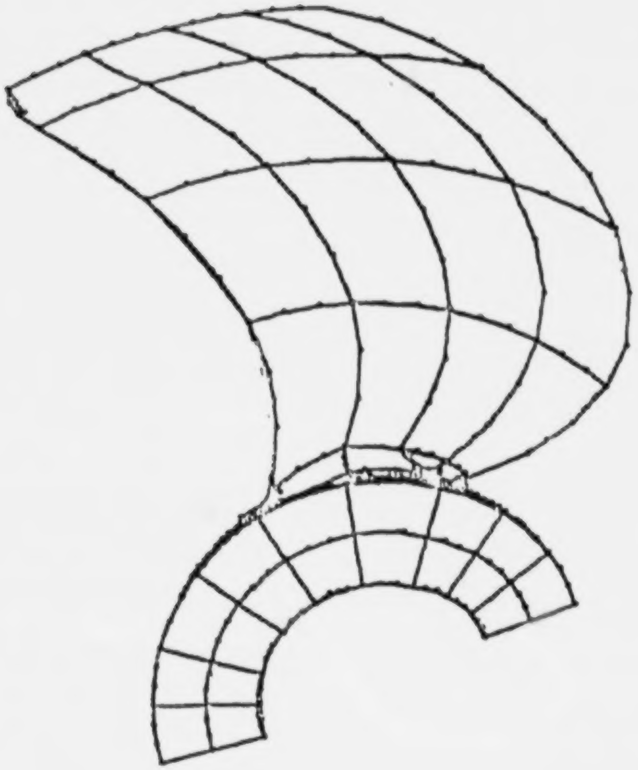


FIGURE 4.4: Substructured Blade and Hub Segment Model

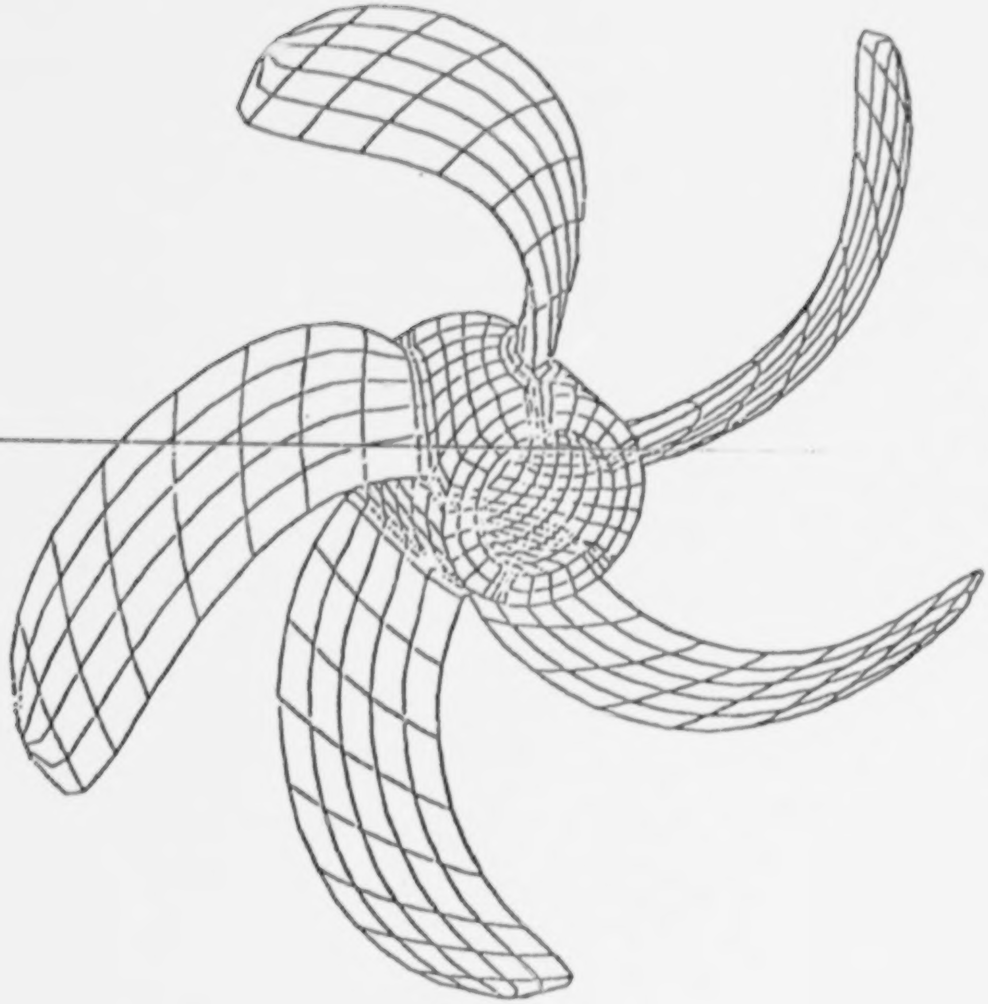


FIGURE 4.5: Assembled Fixed Pitch Propeller Blade

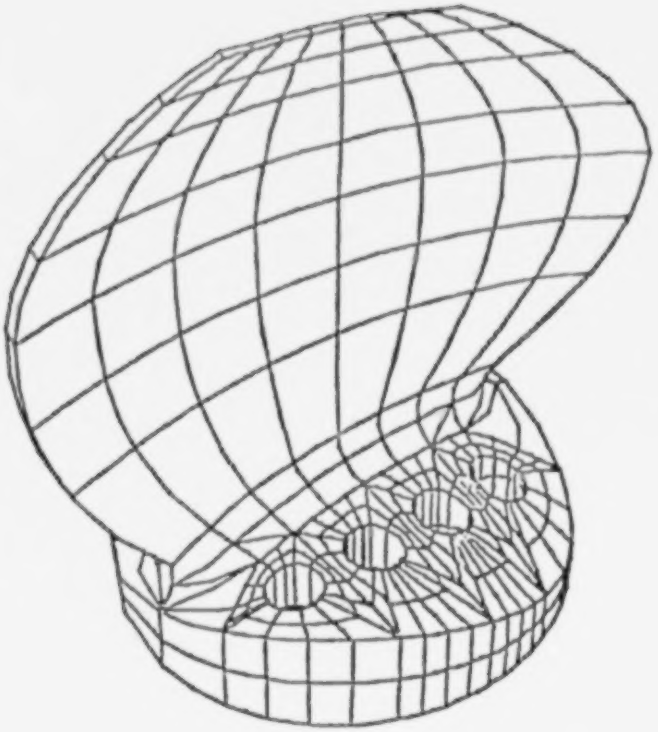


FIGURE 4.6: Detailed Model for Analysis of Attachment Bolts

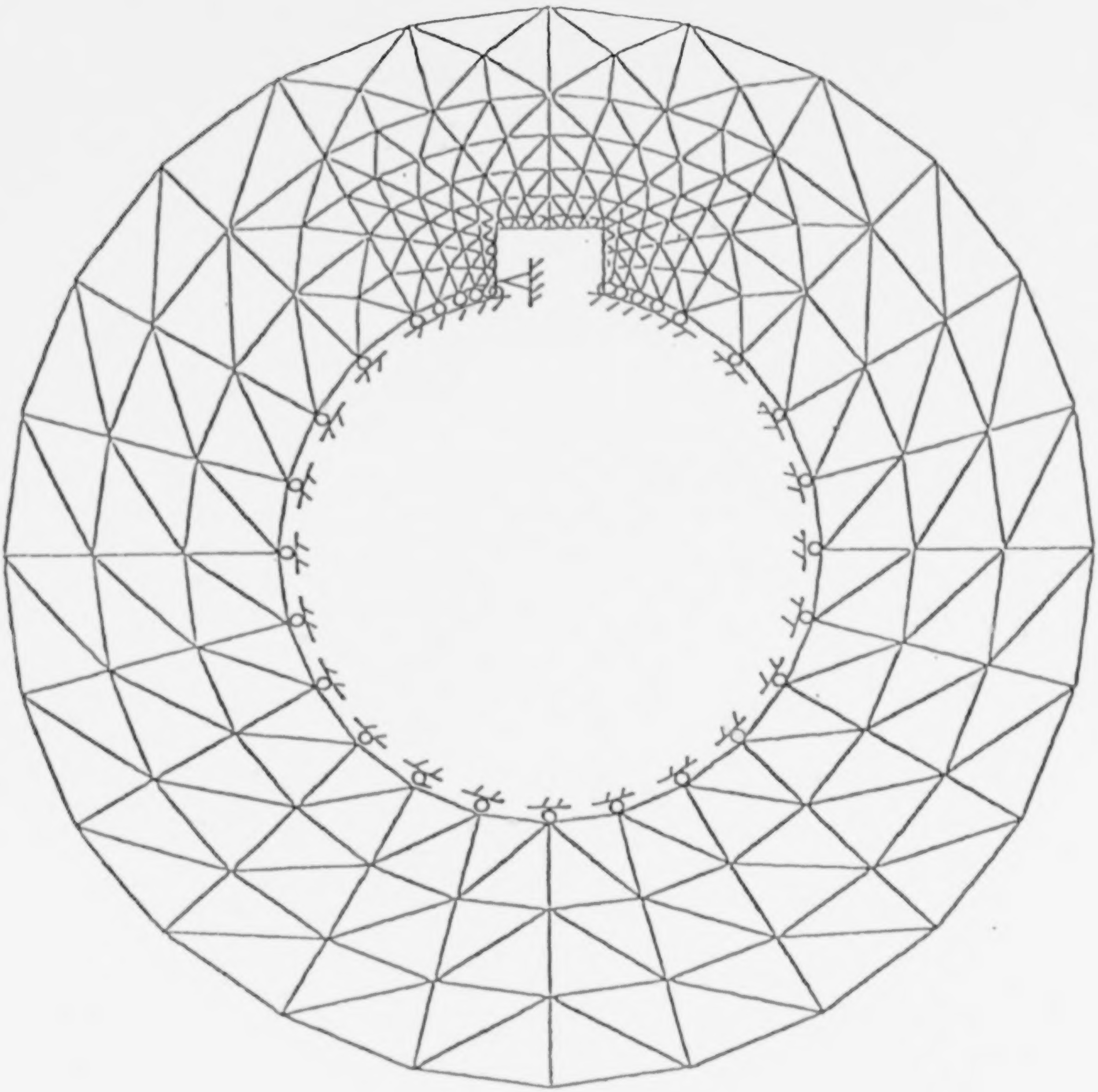
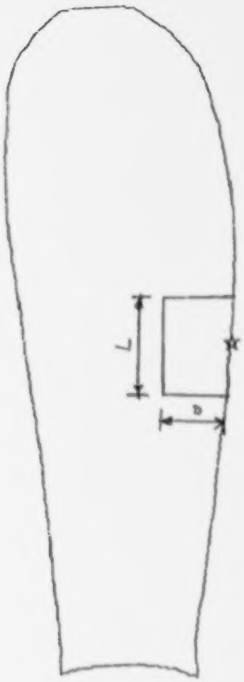
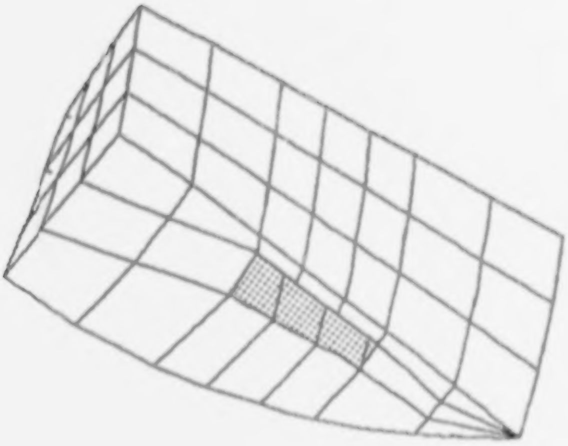


FIGURE 4.7: Hub Keyway Model



(a) Full Blade Model



(b) Top-Down Air Channel Model

FIGURE 4.8: Model Air Channel Effects

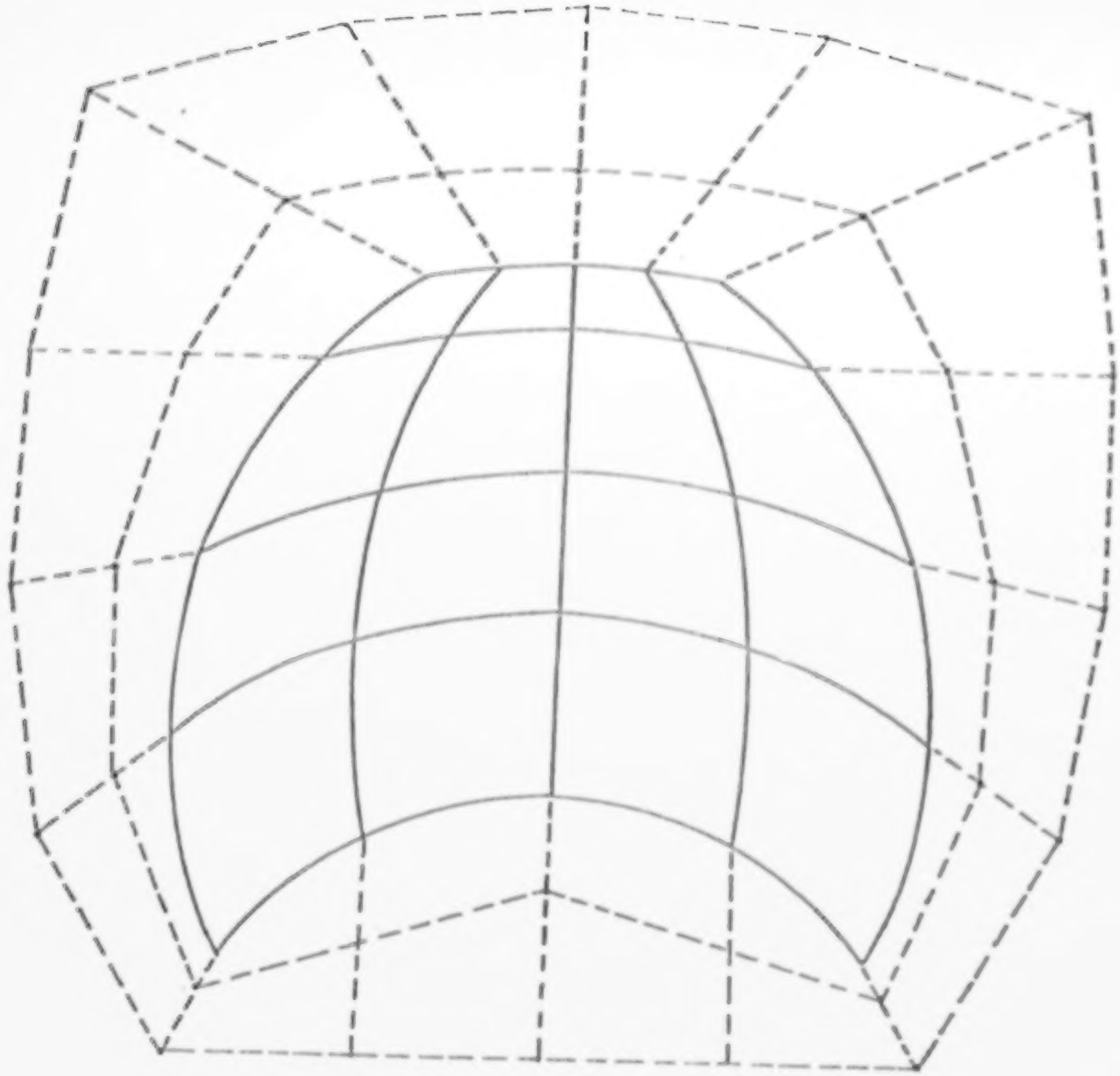


FIGURE 4.9: Finite Element Mesh for a Typical Propeller-Fluid Model

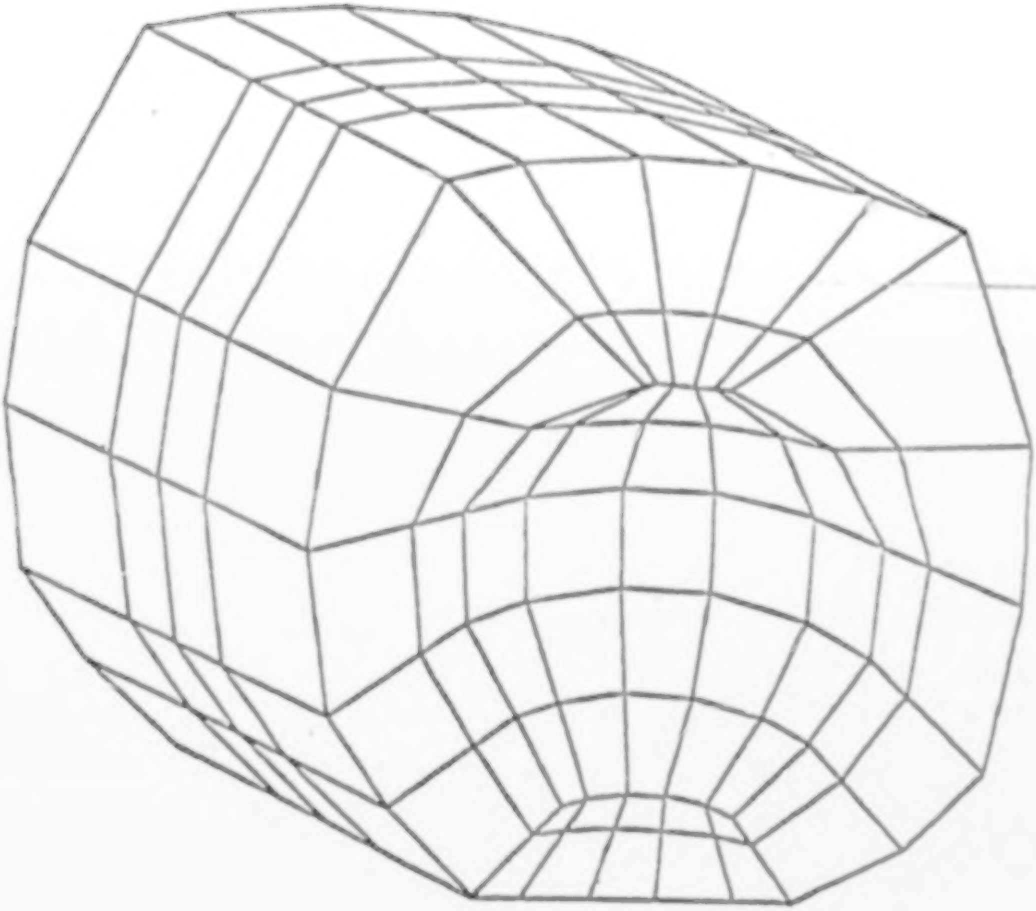


FIGURE 4.10: Finite Element Model of Fluid Surrounding Blade

5.0 PROGRAM ORGANIZATION AND OPERATION

5.1 General

As stated in Section 1, the analysis of propellers is divided into three phases: (i) pre-processing, (ii) solution, and (iii) post- processing. The solution phase is handled by the program VAST. The pre- and post-processing phases are handled by general and special purpose auxiliary programs within the PVAST system.

VAST is the nucleus of the propeller analysis system. The basic features of VAST are summarized in Table 5.1. Also offered with the VAST program are a number of general purpose auxiliary programs which provide: (i) interactive graphics for plotting finite element models, (ii) interactive graphics for plotting results, and (iii) a finite element procedure for computing hydrodynamic added mass (ADMAS1 to ADMAS4). The functions of these programs are listed in Table 5.2.

PVAST contains a special package of pre- and post-processing designed solely for propeller analysis. This program package consists of 25 sub-programs which operate essentially independently of each other, with pertinent data being transferred between them via disk storage. A list of the programs and the basic functions of each are provided in Table 5.3 and 5.4.

Two different modes for operating, the pre- and post-processing programs are available. When operating Mode #1 (MODE=1), the individual programs are loaded and executed, one at a time, and in the proper sequence, as shown in Figures 5.1-1 and 5.1-2. When operating in Mode #2 (MODE=2), the various programs are organized as a series of overlaid programs loaded as a single pre-processing program, and a single post-processing program. The overlaid pre-processing program is called PVAST4 and is described in Appendix K. The overlaid post-processing program is called PPOST4 and is described in Appendix L. When using PVAST4 and PPOST4, desired options are selected by interactive questions and answers.

5.2 Input/Output

Data input in the form of cards, files and/or terminal keyboard responses to prompts are described in Appendices A-K.

Both VAST and PVAST use disk files for out-of-core storage and transferring information from program to program. Some of these are scratch files and are generally automatically deleted after program execution. Others are lineprinter output files which are to be printed and/or deleted at user discretion. The remainder are permanent files which are not to be deleted until all programs which they serve as input have been successfully executed.

The PVAST programs generally identify disk files as PREFX.XXXX where the extension .XXXX comprises of one or two alpha characters followed by one or two numeric characters. Note that extensions beginning with the letter S are associated with scratch files and those beginning with the letters LP are associated with lineprinter output files. PREFX is a unique five character alphanumeric file name prefix assigned to the current propeller analysis by the user to prevent undesirable interactions with files of another independent analysis. Summaries of I/O disk files for the PVAST programs are provided in Tables 5.5 and 5.6. The input data to the pre-processing program PVAST4 is generally provided on the file .CDR or interactively from the keyboard.

The VAST program similarly identifies its disk files as PREFX.TXX or PREFX.SXX where extensions .TXX indicate permanent files and .SXX scratch files.XX are file numbers. The input data for VAST goes on the data set files PREFX.USE, PREFX.GOM and PREFX.LOD.

TABLE 5.1

Features of the Program VAST

- Wide selection of structural elements

- Substructures (static and natural frequency analysis only)
  - Constrained substructures
  - Mirrored and rotated substructures

- Bandwidth optimization

- Natural frequency and mode shape analysis:
  - In air or water
  - Constrained or free-free
  - Centrifugal stiffening
  - Direct iteration method
  - Subspace iteration method

- Static analysis (displacements and stresses):
  - Concentrated loads
  - Distributed external loading
  - Distributed internal loading (rigid body accelerations and rotational velocities and accelerations)

- Dynamic analysis (displacements and stresses):
  - Concentrated loads
  - Distributed external loading
  - Ground motion
  - Solution via modal superposition
  - Solution via direct integration

- Response spectrum analysis (displacements and stresses):
  - Displacement vs frequency or acceleration vs frequency
  - ABS, RMS, or PRMS displacements and stresses

- Frequency response analysis:
  - Sinusoidal loads
  - Periodic loading based on Fourier Series representation

TABLE 5.2

VAST Auxiliary Programs

Pre-Processing Programs

PLOTV1 -	Interactive graphics program for plotting finite element models and for verifying applied boundary conditions.
PLOTHD -	Program for interfacing VAST and MOVIE.BYU, a graphics program for displaying three-dimensional perspective drawings of finite element models with hidden line removal. The user manual for MOVIE.BYU is provided in Reference 10.

Added Mass Programs

ADMAS1 -	Generates fluid and interface element matrices.
ADMAS2 -	Assembles fluid and interface element matrices.
ADMAS3 -	Decomposes assembled fluid matrix.
ADMAS4 -	Generates added mass matrix.
ADMAS5 -	Generates added mass matrix for the surface panel method.

Post-Processing Programs

POSTV2 -	A non-plotting post processing program which generates a stress data file in the format required by various stress plotting programs from VAST stress output.
POSTV3 -	A non-plotting post processing program which generates a stress or displacement function data file for PLOTHD using either mode shapes, deflections or stresses.
PLOTV2 -	Interactive graphics program to plot displaced shapes due to static or dynamic loading, or mode shapes from a natural frequency or buckling analysis. Distorted finite element grids are superimposed on the undistorted finite element grid. In the case of a dynamic analysis, displaced shapes at any specified time step can be obtained.
PLOTV3 -	Interactive graphics program to plot stress contours for static and dynamic analyses. Contours can be obtained for principal stresses or von Mises-Hencky yield stress as well as basic stress components (up to six depending on element type). In the case of a dynamic analysis, contours at any specified time step can be obtained, or, the time of occurrence of the maximum stress can be requested, and the corresponding contour would then be plotted.
PLOTV4 -	Interactive program to plot principal stress vectors for static and dynamic analyses. In the case of dynamic analyses, stress vectors at any specified time step can be obtained.

PLOTV5 - Interactive program to plot time histories of structural response as displacement versus time at specified nodal points.

PLOTV6 - Plots frequency response in the form of amplitude ratios versus forcing frequency at specified nodes.

PLOTV7 - Plots time histories of structural response as stress versus time at specified nodal points. Up to six stress components (depending on element time), principal stresses and the von Mises-Hencky yield stress can be obtained.

PLOTV8 - Interactive graphics program for fatigue failure assessments of stresses from a dynamic response analysis. Range and mean equivalent von Mises uniaxial nodal stresses are plotted on a combined Soderberg, Goodman, Gerber, Ellipse, SAE and Bagci diagram. At user selected nodal points, cycles-to-failure due to fatigue, stress safety factors and time histories of equivalent von Mises uniaxial stresses may also be displayed.

PLOTV9 - Plots displacement contours from a static or dynamic loading, or from a natural frequency or buckling analysis. In the case of dynamic analysis, contours at any specified time step can be obtained. In the case of a natural frequency or buckling analysis, contours of any specified node can be obtained.

PLTV10 - Interactive graphics program to plot superelement displaced shapes due to static or dynamic loading, or mode shapes from a natural frequency or buckling analysis. Distorted superelement outlines are superimposed on the undistorted superelement outlines. In the case of a dynamic analysis, displaced shapes at any specified time step can be obtained.

PLTV12 - Interactive graphics program for plotting applied pressure loads or concentrated forces and moments acting on structural finite element models.

PLOTHD - Program for interfacing VAST and MOVIE.BYU, a graphics program for displaying three-dimensional perspective drawings of mode shapes, deflections or stresses with hidden line removal. The user manual for MOVIE.BYU is provided in Reference 10.

TABLE 5.3

PVAST Pre-Processing Programs

BLADAT - Permits user to interactively input and edit several standard data forms for describing basic propeller blade section geometry including:

(1) Ratios of half thickness over maximum thickness and camber over maximum chamber.

(2) NACA section data, for the basic section.

(3) Section offsets from drawings.

BLADGM - Generates data files for processing and plotting from propeller blade section data supplied in one of the standard forms. In particular, the program produces a tabular lineprinter output of three dimensional blade coordinates; a formatted data file of the three dimensional coordinates suitable for input to programs PVBLD1 of PVAST; and unformatted tabular data files from which graphics of the blade are plotted using program BLADPL.

BLADPL - Plots basic propeller blade data generated by program BLADGM.

(1) Graphs of input data.

(2) Graphs of output data.

(3) Plots of magnified nose/tail of blade sections.

(4) Three-dimensional plots of blade and assembled propeller.

(5) Plots of individual sections.

(6) Plots of maximum thickness (skew removed).

PVBLD1 - (i) Calculates the true-space coordinates of propeller blades using the designers basic blade design data from BLADGM.

(ii) Calculates center of gravity and weight for a single blade.

(iii) Calculates expanded and projected area of a single blade.

(iv) Calculates rotary inertias of a single propeller blade.

(v) Calculates translational and rotational components of the added mass for 3, 4, and 5 bladed propellers using empirical methods.

(vi) Accepts the finite element grid selected for the propeller blade.

PVFIL - Generates finite element models of the fillet transitions between propeller blades and their palm or hub bases (using data generated by PVBLD1). Flexibility in fillet geometry is manifested through options for single radius, compound radius or continuous function (cubic spline) fillet gauge curves. Different fillet gauge definitions may apply for pressure (front) and suction (back) hydrodynamic blade surfaces. Provisions are also made for chordwise variations in fillet gauge parameter definitions. Either two or three elements may be distributed along the fillet curves. Wrap around fillets from the pressure side to the suction side are also provided at both leading and trailing edges.

PVHUB - Generates finite element models of a hub segment for connection to the propeller blade through the fillet generated by PVFIL with improved element proportioning. Elements of the hub extensions near the blade leading and trailing edges, do not progressively degenerate as fillet radii decrease at those edges. Because fillets can be nonsymmetric on either side of the blade chord, provisions have been made to center the hub beneath the fillet. Also in order to control element dimensions in the circumferential direction, an option to produce a bladeless hub segment with an arbitrary number of elements in the circumferential direction is provided.

PVPALM - Generates finite element model of propeller palm.

PVBLD2 - Generates one of three types of finite element models for the propeller blade using data generated by PVBLD1 and PVFIL.

(i) First radial position of geometric data taken as blade base for analysis of blade alone;

(ii) Fillet data produced by the fillet modelling program PVFIL used to model a fillet with one element through the thickness providing more realistic representation of blade root boundary conditions; and

(iii) A substructure of blade starting at the top of fillet required when palm or hub attachments are contemplated.

PVBLD3 - Provides capability to interactively describe finite element model for blade via transverse view drawing using a graphics tablet with increases in user control over element proportioning particularly in region of blade tip.

PVSUB1 - Generates substructure/superelement models for combined blade, fillet and palm or hub structures in single or multi-blade applications (using data generated by subprogram PVFIL, PVBLD2, and PVHUB or PVPALM). Two element bolt substructures are also optionally included for palm bases.

PVSUB2 - Generates substructure/superelement models for combined blade, fillet and palm structures when bolt holes and bolts are modelled in detail as substructures (using data generated by subprograms PVFIL, PVBLD2, PVPALM and PVBOLT).

PVFLU1 - Converts three-dimensional fluid model data to single layer thin fluid model and allows user to interactively modify the radius, chord, skew, rake and pitch angle data of the fluid model.

PVFLU2 - Generates finite element model for the fluid surrounding propeller blade.

PVFLU3 - Generates surface panel model for the fluid surrounding propeller blade.

PVKEY - Generates two dimensional finite element models of propeller hub containing keyways and their loads.

PVCHL1 - Generates air channel model and creates a PREFIX.USE file for detailed analysis of the air channel.

PVLODA - Adjusts applied propeller pressure loading to reduce pressure spikes along the leading and trailing edges from which loading errors develop. The smoothed chordwise pressure distributions produce the same thrust and spindle torque as the original pressure distribution.

PVLODB - (i) Transforms propeller blade pressure and drag loading datasets into form acceptable to PVLODV.

(ii) Calculates thrust, spindle torque and shaft torque generated by a single blade subjected to the loads defined on each dataset.

PVLODV - Generates load data for VAST associated with combined element pressures, centrifugal loads, drag forces and bolt prestrains where consistent nodal point forces will be computed for the finite element model of complete propeller blade structures.

PVBCON - Provides capability for interactive input and/or modification of boundary conditions for the finite element model.

PVGAP - Provides the capability to consolidate into a single file, the gap element information for the finite element model defined automatically by programs or optionally defined interactively by user in earlier program steps which is generally stored on several independent data files with options for final interactive modification.

PVBLTA - (i) Provides the capability to interactively relocate nodes on the propeller palm base so as to achieve suitable bolt attachment points.

(ii) Provides the capability to identify nodes on the propeller palm base to which gap elements will be attached or boundary conditions applied.

PVBLTB - (i) Provides the capability to interactively relocate nodes on the propeller palm base to achieve suitably shaped elements for replacement with bolt hole substructures and connecting bolt substructures.

(ii) Provides the capability to identify nodes on the propeller palm base to which gap elements will be attached or boundary conditions applied.

PVBLTC - Provides the capability to identify nodes on the base of each bolt hole substructure to which gap elements will be attached or boundary conditions applied.

PVBOLT - Generates bolt and bolt hole substructure pairs and produces required superelement data including multi-point constraint equation information as well as default boundary conditions and gap element data.

PVUSE - Generates .USE file and submit control file .CTL for analysis.

PGBLAD - Provides the following propeller blade plots: (1) expanded outline; (2) developed sections; (3) developed outline; (4) transverse projection; (5) longitudinal projection; (6) plan view; and (7) finite element grid and node points.

PGLOD1 - Provides plots of input and adjusted chordwise distributions of pressure loading. For dynamic loading, time history plots of pressure loading at the specified locations can be displayed.

PGLOD2 - Provides chordwise plots of input, adjusted and interpolated grid point pressure loadings, as well as table of finite element grid thrust, and shaft and spindle torques due to pressure and drag loadings.

PGFEM - Plots finite element models of the blade, fillet, palm and hub.

PGFLUD - Plots fluid finite element models.

TABLE 5.4

PVAST Post-Processing Programs

PVDS1 - (i) Processes VAST displacement and stress results from static analysis to be used by PGSTR1/2/3 and PGDIS2/3.

(ii) Prints tabulated global displacements; normal, tangential, and radial displacements, and principal, tangential, and radial stresses.

PVSD2 - (i) Processes VAST displacement and stress results from dynamic analysis to be used by PGDIS1.

(ii) Prints tabulated global displacements; normal, tangential and radial displacements; and principal, tangential, and radial stresses.

PVFOR1 - (i) Processes force vector data and computes normal, radial and tangential force components for plotting purposes by PGFOR1.

(ii) Prints tabulated global forces as well as transformed normal, tangential and radial components.

PVSECT - Provides a capability to identify finite element surfaces on either selected horizontal or vertical propeller blade cross-sections on which stress contours are to be plotted using PLOTV3.

PVFOR2 - Computes from the results of a PVAST dynamic analysis for a single propeller blade (operating in a wake), the time histories of force and moment resultants acting on the shaft for a single blade or for the whole (multi-bladed) propeller.

PGDIS1 - Plots deflected blade shapes resulting from static loading and/or natural mode shapes from a free vibration analysis.

PGDS1 - Provides, at specified radius and chord, the time histories of the following: global X, Y, and Z displacement; normal, tangential, and radial displacements; principal stresses; and tangential and radial stress components.

PGSTR1 - Provides, at specified radius, chordwise distributions of principal stresses, and radial and tangential stress components (static analysis only).

PGDIS2 - Provides, at specified radius, chordwise variations of normal, tangential, and radial displacement components (static analysis only).

PGDIS3 - Provides, at specified radius, the fully developed view of the blade section under static load superimposed on the unloaded section to show section distortion.

PGSTR2 - Provides primary and secondary principal stress contours (static analysis only).

PGSTR3 - Plots principal stress vectors for propeller blade (static analysis only).

PGDIS4 - Provides plots, at specified radius, of natural frequency mode displaced blade section superimposed on the unloaded blade section.

PGDIS5 - Plots displacement contours of mode shapes of propeller blade.

PGFOR1 - Plots radial, tangential and normal force vector components.

PGFOR3 - Plots time histories of input pressures, finite element pressures and drag forces.

PGFOR2 - Plots time histories of resultant shaft forces and moments for a single blade or whole (multi-bladed) propeller.

TABLE 5.5

Summary of I/O Disk Files for PVAST Pre-Processing Programs

Program Name (.SR4)	File Name Extension[1]		
	In	Out	Scratch
PVEDIT	CDR	CDR	S61,S62,S63,S64,S65
BLADAT	--	DAT	--
BLADGM	DAT	T16, POD, PPD, PMN, PSP, PPR, PTH, LBP	--
BLADPL	POD,PPD,PMN,PSP,PPR,PTH	--	--
PVBLD1	CDR,T16[2]	T11, LP1[1]	--
PVBPI	CDR,T11	--	--
PVFIL	CDR,T11	ROT, FIL, LP2A	S61,S62,S63,S64
PVHUB	ROT,FIL	HUB[5], LP2B	--
PVPALM	ROT,FIL	PLM[5], T28, LP2C	--
PVLBD2	CDR,T11,FIL[3]	T10[8], LP3	--
PVBLD3	T11	T10[8]	--
PVSUB1	CDR,T10,T11,FIL,HUB,PLM,T28	T27[9], LP5A	--
PVSUB2	CDR,T10,T11,FIL, PLM, T13, BHO	T27[9], LP5B	S61,S62,S63
PVFLU1	CDR,T11,CFM	CFM	--
PVFLU2	CDR,CFM,T10,T11	T30 or THN[11], LP6	S61
PVFLU3	T10	AMD	--
PVKEY	CDR	T20[4], LOD	S61
PVCHL1	CDR,T10,T11,LOD,T52	CHL	--
PVLODA	LIN	LAD	--
PVLODB	T11 AND CDR, LIN OR LAD[10]	T12, LP8	--
PVLODV	CDR, T11, T12, T28	LOD, LP9	S61,S62
PVBCON	TBC or (USE and T10, T20, T27, or PLM and BHO)	TBC, LP10	S61,S62,S63
PVGAP	GAP or (USE and T10, T20, T27, or PLM and BHO)	GAP, LP10	S61,S62,S63
PVBLTA	CDR[6], ROT, PLM[7]	T28	S61,S62
PVBLTB	CDR[6], PLM[7], ROT	T13	S61,S62,S63

TABLE 5.5 (Continued)

Summary of I/O Disk Files for PVAST Pre-Processing Programs

rogram Name (.SR4)	File Name Extension[1]		
	In	Out	Scratch
BLTC	BHO	BHO[12]	S61,S62,S63
USE	T10, T27, T20, TBC, GAP	USE, CTL	--
OLT	CDR, ROT, T13, PLM	BHO, LP18	S61, S62, S63, S64, S65, S66
LAD	T11	--	--
LOD1	LIN, LAD	--	--
LOD2	T12	--	--
EM	T10, or T17	--	--
OD3	T12	--	--
LUD	T30	--	--
TV1	FIL, PLM, HUB, BHO, T10, T27, SUP, TBC, USE, T20	--	S31, S32, S33, S34
THD	FIL, PLM, HUB, BHO, T10, T27, THN, T30	MOV	S27,S28,S30,S31, ETC.[13]
V4	MOV	--	--

NOTES:

1. Disk files are identified by names of the form PREFX.XXXX where the extension generally comprises of one or two alpha characters followed by one or two numeric characters. Extensions beginning with the letter S refer to scratch files. Those beginning with LP refer to disk files intended for lineprinter output, copies of which can be obtained when user requests printing of files on the lineprinter. All others are permanent files.
2. Optional (alternate source of blade data, instead of cards).
3. Optional (required when IBASE = 1).
4. Contains geometry and superelement data for keyway model.
5. PREFX.GOM file for palm/hub analysis only.
6. Optional (alternate source of bolt data, instead of through interactive prompts via terminal keyboard).
7. Original tape is replaced if palm model has been modified.
8. PREFX.GOM file for blade analysis only (no substructuring).
9. PREFX.GOM file for substructured blade and palm/hub analysis.
10. LAD file is used if available, otherwise LIN.
11. Contains fluid element data for compressed (thin) fluid model.
12. Contains bolt and gap element data.
13. Number of scratch tapes used depends on the number of model superelements.

TABLE 5.6

Summary of I/O Disk Files for PVAST Post-Processing Programs

Program Name (.SR4)	File Name Extension[1]		
	In	Out	Scratch
DS1	T10, T11, T52, T53	T15, LP11	--
DS2	T10, T11, T52, T53	T18, LP12	--
FOR1	T11, T41, T47	T14, LP13	--
SECT	T11, T17[4], T27[4]	PV2, LP14	--
FOR2	CDR, T10, T11, T48, T52	TFR, LP19	--
DIS1	T10, T11, T51[2], T52[3], or T59[4]	--	--
DS1	T11, T18	--	--
STR1	T11, T15	--	--
DIS2	T11, T15	--	--
DIS3	T11, T15	--	--
STR2	T11, T15	--	S61, S62
DIS4	T10, T11, T51	--	--
DIS5	T10, T11, T51	--	S61, S62
STR3	T11, T15	--	--
FOR1	T11, T14	--	--
LOD3	T12	--	S61
FOR2	TFR	--	--
THD	T30	--	--

ES:

ermanent input and output files are identified by the name PREFX.TXX and temporary scratch files y the name PREFX.SXX. PREFX is an unique five character alphanumeric file name prefix assigned an analysis by the user to prevent undesireable interactions with files of another independent analysis.

equired only for plotting mode shapes.

equired only for plotting static deflected blade shape.

equired only when substructuring is used.

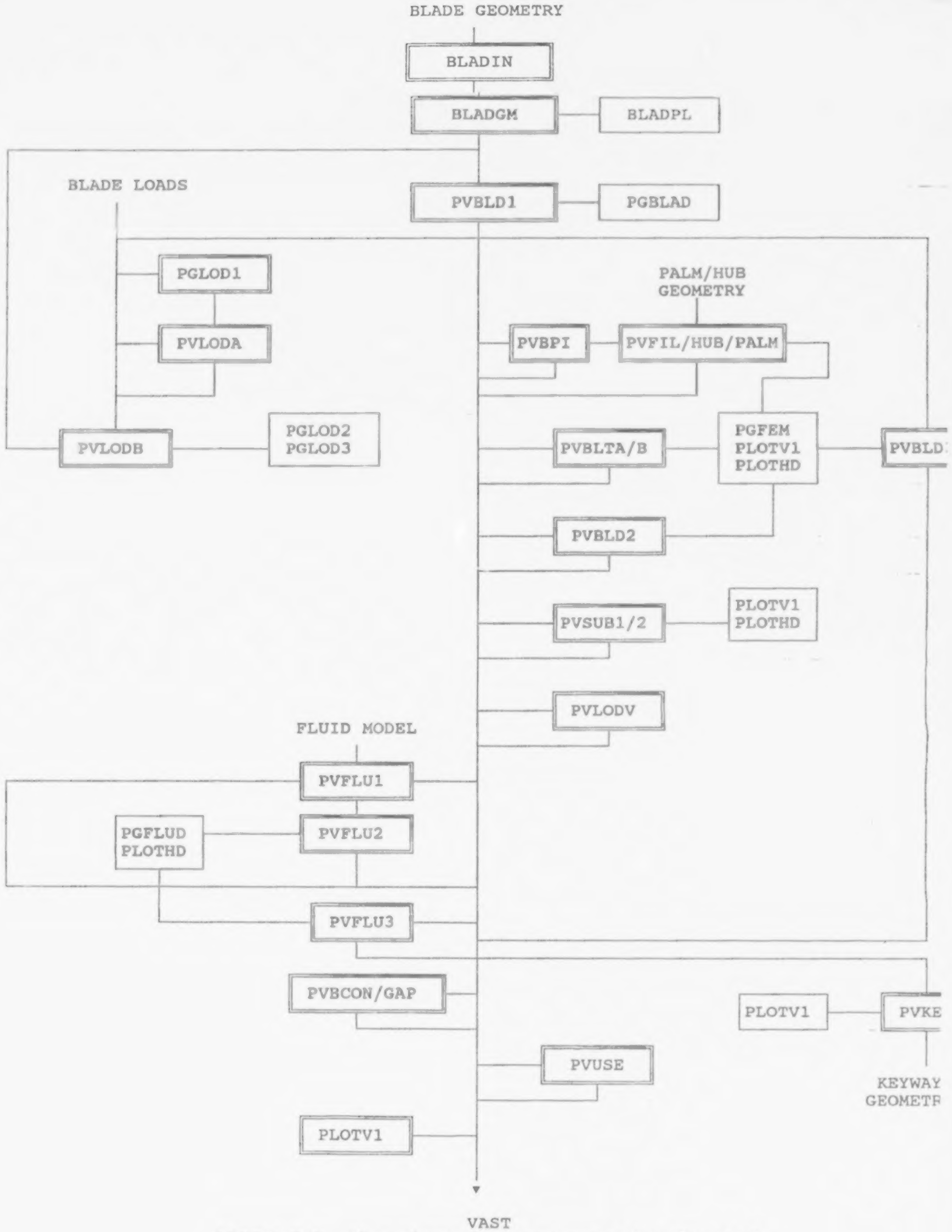

BLADE GEOMETRY
BLADIN
BLADGM
BLADPL
BLADE LOADS
PVBLD1
PGBLAD
PGLOD1
PALM/HUB
GEOMETRY
PVLODA
PVBPI
PVFIL/HUB/PALM
PVLODB
PGLOD2
PGLOD3
PVBLTA/B
PGFEM
PLOTV1
PLOTHD
PVBLD
PVBLD2
PVSUB1/2
PLOTV1
PLOTHD
PVLODV
FLUID MODEL
PVFLU1
PGFLUD
PLOTHD
PVFLU2
PVFLU3
PVBCON/GAP
PLOTV1
PVKE
PVUSE
KEYWAY
GEOMETR
PLOTV1
VAST

FIGURE 5.1: Flow Chart for Pre-Processing Programs

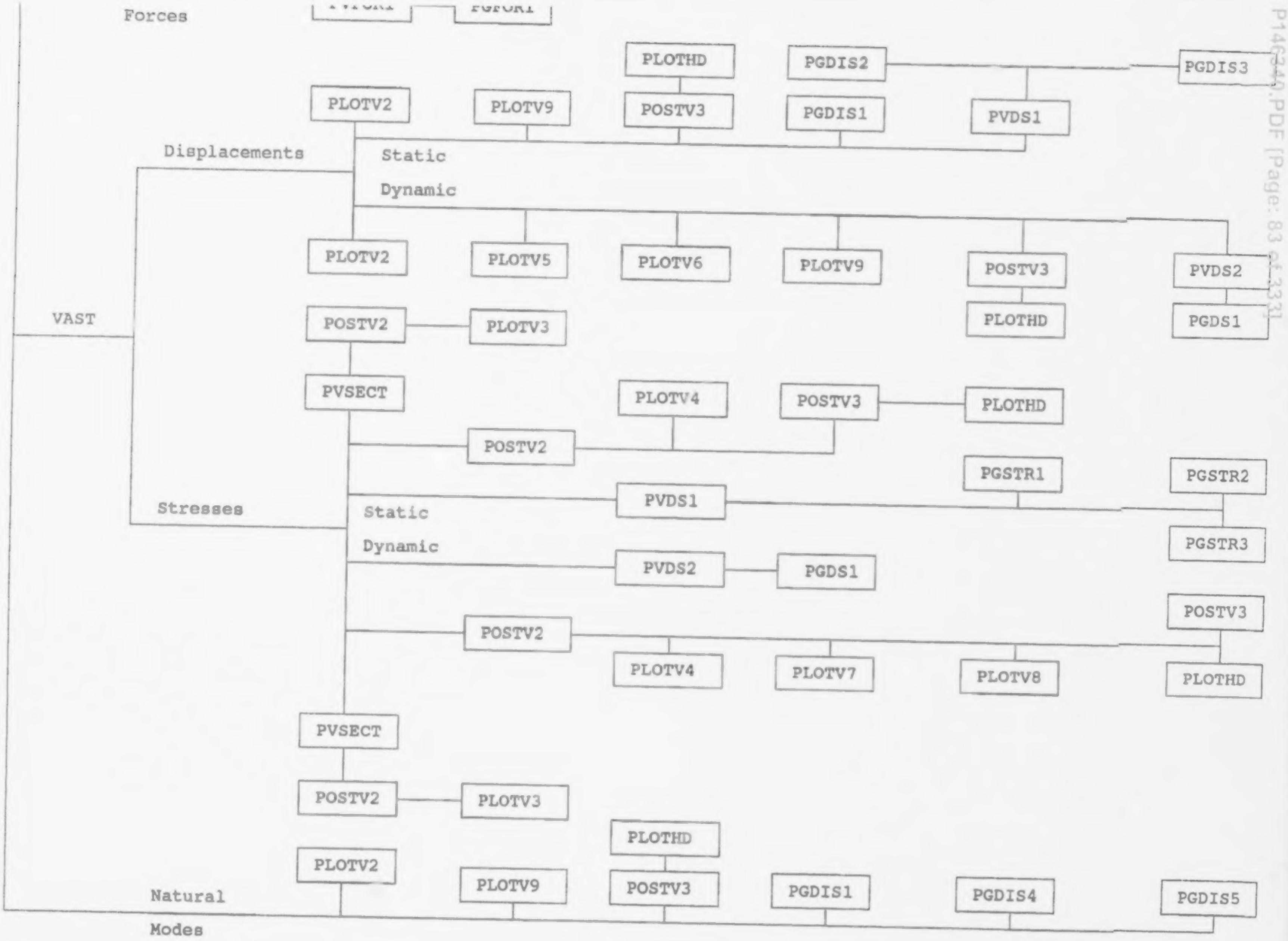

VAST
Forces
Displacements
Static
Dynamic
Stresses
Static
Dynamic
Natural
Modes
PLOTHD
PGDIS2
PGDIS3
PLOTV2
PLOTV9
POSTV3
PGDIS1
PVDS1
PLOTV2
PLOTV5
PLOTV6
PLOTV9
POSTV3
PVDS2
PLOTHD
PGDS1
POSTV2
PLOTV3
PVSECT
PLOTV4
POSTV3
PLOTHD
POSTV2
PGSTR1
PGSTR2
PVDS1
PGSTR3
PVDS2
PGDS1
POSTV3
POSTV2
PLOTV4
PLOTV7
PLOTV8
PLOTHD
PVSECT
POSTV2
PLOTV3
PLOTHD
PLOTV2
PLOTV9
POSTV3
PGDIS1
PGDIS4
PGDIS5

FIGURE 5.2: Flow Chart for Post-Processing Programs

6.0 INPUT PREPARATION USING PRE-PROCESSING PROGRAMS

6.1 Introduction

It is essential that the user of the PVAST pre-processing programs have a basic understanding of propeller geometry. A detailed description of the geometry of marine propellers is provided in Appendix M of this manual. Users with little or no background in propeller geometry should study Appendix M in detail. Users well knowledged in propeller geometry should read Appendix M to become familiar with such aspects as coordinate systems, sign-conventions, and terminology.

The pre-processors in PVAST can be grouped into three categories: (i) those dealing with structural model generation; (ii) those dealing with fluid model generation (i.e. the water surrounding the blade); and (iii) those dealing with load generation.

6.2 Structural Finite Element Models

The co-ordinate system used in this manual to describe the true space geometry is shown in Figure 6.1. The right-handed Cartesian co-ordinate system XYZ is oriented so that the Z axis is the axis of propeller rotation and is positive in the aft direction. When looking forward, right-handed propellers are assumed to be rotating clockwise. The Y axis coincides with the vertical axis of the blade. Each developed section is defined by a local co-ordinate x'z' and its pitch angle, as shown in Figure 6.2. By a series of co-ordinate transformations, the co-ordinates in the XYZ system can be obtained.

PVBLD1 is the primary pre-processor and is supported by the plotting program PGBLAD. It accepts the designer's basic blade design data and converts it into the true space XYZ co-ordinate system, which is acceptable to the finite element programs. PGBLAD can be used for plotting: expanded outlines, developed sections, developed outlines, plan views, transverse projections, longitudinal projections, and finite element grid and nodal points. Typical plots from PGBLAD are shown in Figure 6.3.

PVBLD1 requires as input section data, at a total of K radial positions, the front and back section offsets for the true developed sections - with respect to the local x'z' co-ordinate system - at L specified chordwise positions, plus the base chord line pitch angles, skew and rake. The last section of the model is treated as a standard section and must be located at a radial position which is slightly smaller than the full blade radius of the actual tip. Two options are available for specifying the chordwise positions for defining face and back section offsets. If ICHORD=1, chordwise positions for the first section are specified, and those for the remaining K-1 sections are assumed to be identical. If ICHORD=2, chordwise positions are specified for each of the K sections. (ICHORD is an input parameter for PVBLD1.)

The K radial positions and the L chordwise positions represent a grid of K x L points. The XYZ co-ordinates for the back and face surfaces of the blade at these points are computed.

The desired number of finite element grid divisions in the radial and chordwise directions (NEK and NEL) must be specified by the user. Also, NEK+1 fraction radii and NEL+1 fraction chord positions defining the gird lines must be supplied. Various options are available for defining these fractions, including a grid biasing feature. These fractions are used to locate the corner nodes of the finite element model, mid-side nodes are located automatically at the approximate centres of the sides of the elements. Altogether, NEK x 2+1 (or KP) radial positions and NEL x 2+1 (or LP) chordwise positions are established.

BLADIN, BLADGM and BLADPL are optional auxiliary programs for PVBLD1. BLADIN permits the user to interactively provide geometry data of blade. BLADGM generates data files for processing and plotting from propeller blade data in one of several basic forms (see Table 5.3). It creates a data file suitable for input to program PVBLD1. BLADPL plots propeller data supplied to and generated by BLADGM.

PVFIL, PVHUB, PVPALM, PVBLD2, and PVBLD3 use the converted data from PVBLD1 to generate finite element meshes and node point co-ordinates for the palm/hub and blade finite element models. Subprograms PVFIL, PVHUB, PVPALM generate the fillet, palm and hub models, respectively, while both PVBLD2 and PVBLD3 generate blade models. PVBLD2 and PVBLD3 are supported by the program PGFEM for plotting blade finite element models. The VAST graphics programs PLOTV1 and PLOTHD can also be used for plotting these models.

PVSUB1 and PVSUB2 use the root and blade models generated from PVFIL, PVHUB, PVPALM and PVBLD2 to combine the blade and root using the substructure technique. The VAST graphics programs PLOTV1 and PLOTHD can be used to plot combined blade and root models.

From the data supplied to PVBLD1, PVBLD2 generates a square or rectangular finite element grid, i.e. square or rectangular with respect to the number of subdivisions in the radial and chordwise directions, not geometry. It uses points dictated by the KP radial positions and LP chordwise positions as locations for the nodes of the finite element model see Figure 6.4). The first, third, and fifth, etc. radial and chordwise positions form the corner nodes of the finite element mesh, while the second, fourth, etc. radial and chordwise positions form the mid-side nodes. However, this pattern is disrupted slightly at the leading and trailing edges near the blade tip when the user requires optimization of the finite element grid at the tip. This is discussed in detail later in this section. PVBLD3, on the other hand, provides the capability to interactively describe the finite element grid for the blade via a transverse view drawing using a graphics tablet. Instructions for using PVBLD3 are given in Appendix B.4. PGFEM, PLOTV1 and PLOTHD can be used to plot hand generated models.

PVBLD2 provides the option to generate one of three blade models. The first, obtained by setting the parameter IBASE=0, assumes a rigid root, the boundary being defined along a specified constant radius. The increase in blade thickness due to fillet material is not included. The second, obtained by setting IBASE=1, offers a more detailed blade model by using node point co-ordinates of the fillet model generated by PVFIL to form an additional row of elements in the chordwise direction, which contains the root fillet. This model offers a means for taking into account root flexibility. However, stresses computed in this area should be interpreted with caution, since only one element is used through the thickness. The third option, obtained by setting IBASE=2 generates a blade model for substructuring. The first row of elements starts at the top of the fillet.

The blade can be modelled using either thick/thin shell elements or 20-noded solid elements. These elements cannot be combined. If shell elements are being used and a model for substructuring is being generated (i.e. IBASE=2) the first row of elements at the bottom of the blade are replaced by transition elements.

When IBASE=0 or 1, one of two node numbering schemes can be specified for the blade model. Setting the flag IGRID=1, the nodes are numbered as shown in Figure 6.5(a); setting the flag IGRID=2, the nodes are numbered as shown in Figure 6.5(b). This feature is useful for stiffness matrix bandwidth minimization. To minimize the bandwidth, number in the direction which has the least number of points in the radial (Grid #1) or chordwise (Grid #2) direction. If IBASE=2 (i.e. substructuring), IGRID must be equal to 2.

An additional feature provided by PVBLD1/2 enables the user to modify the finite element grid pattern to achieve well proportioned elements throughout the blade. This finite element grid optimization can be achieved by either or both of two methods. The first method has to do with the positioning of the finite element grid points while the second has to do with a slight modification of the elements at the leading and/or trailing edges near the blade tip. See Reference [13] for details.

In the first method, the user chooses the option to define the NEK+1 fraction radii himself. Then he is prompted for the number of chordwise positions (NRLN) at which these NEK+1 fraction radii are to be specified. If NRLN=1, fraction radii are specified at only one chordwise position (the leading edge), and those for the remaining NEL chordwise positions are assumed to be identical. If NRLN=2, fraction radii are specified at the leading and trailing edges, respectively and those for the other chordwise positions are obtained by linear interpolation. If NRLN=3, fraction radii are specified at the leading edge, trailing edge and mid-chord, in that order, and those for the remaining chordwise positions are obtained by quadratic interpolation. NRLN>1 will prove particularly useful for highly skewed blade situations.

The second method is controlled by the optimization parameter LOPTM. If LOPTM=0, this method is not performed. If LOPTM=1 or 2, the element at the leading or trailing edge, respectively, at the blade tip is automatically modified. If LOPTM=3, the elements at the leading and trailing edges at the blade tip are modified. Figure 6.6 shows a modified grid for the case with LOPTM=3.

When the 20-noded solid elements are used to model the blade it is possible to collapse the elements to wedges at the leading and trailing edges if the blade thickness at these edges is very small. This option is activated by setting the parameter IWDGE=1. If IWDGE=0, wedges are not formed even if the blade thickness at the edges is very small.

Two types of root models are available, i.e. a palm model and a hub model, as shown in Figure 6.7. There are two types of root models for blade intersection. These are the palm and hub models as shown in Figure 6.8. At present the palm surface can be modelled by a single 16-noded patch while the hub surface can be modelled by a cubic spline generator. The fillet radius can be described by a single or compound radius or a spline function continuous up to and including the second derivative and a typical fillet substructure model is shown in Figure 6.9. It should be noted that the fillet model contains two element layers (even if a three element fillet is employed). The parameter NES is used to define the location of the root base end relative to the leading edge of the fillet model (see Figure 6.10). If NES=0, it means that the base begins at the leading edge; if NES=-1, the base is extended one division beyond the leading edge; if NES>0, it means an overhang of the blade at the leading edge of NES divisions. Similarly, the parameter NEF is used to locate the root base end on the trailing edge side. Parameters NES and NEF are redundant in hub model generation as hub extensions are always produced at both leading and tailing edges.

PVFIL uses an iteration process which establishes a circle, containing the fillet, tangent to the blade surface, and approximately tangent to the palm surface. The procedure involves the movement of (i) the circle (at a specified chordwise position) in the radial direction, through increments of DR x FULL BLADE RADIUS, and (ii) a point on the circumference of the circle, through increments of DANG degrees. The accuracy of the procedure depends on DR, DANG, and a tolerance TOL; and the smaller these values are, the higher the accuracy. However, increased accuracy means higher computer costs. PVFIL/HUB/PALM defaults to values of 0.1, 1.0, and 0.001 for DR, DANG, and TOL, respectively. These values appear to be a good compromise between accuracy and computer costs.

PVSUBI provides the user with the following substructure/superelement modelling of the propeller:

1. Combine the blade and root models using the 20-noded isoparametric element.

2. Combine the blade and root models using the blade shell element, the transition element and the 20-noded isoparametric element.

Two types of substructured propeller models are possible: (i) single blade, fillet and palm/hub section, and (ii) all blades, their fillets and a complete hub. These models are shown in Figure 6.11.

PVKEY generates a two-dimensional model of a hub containing the keyway, as shown in Figure 6.12. The model is comprised of four superelements. Superelements #1 and #3 are shown in Figures 6.13 and 6.14. Superelement #2 is a mirror image of superelement #1 and superelement #4 is a mirror image of superelement #3. Skewed coordinates and gap elements are used at the hub-shaft interface. The user provides the inner and outer hub radii, the hub length, the depth and width of the keyway and blade loading. Blade loading is provided in the form of radial and tangential forces as well as moments about the shaft axis. Three outer circumference nodes nearest to a blade are selected (transparent to the user), and the blade force and moment applied to these nodes.

If the option to determine bolt loads by using rod elements is used, PVBLTA must be used. Its purpose is to allow the user to interactively relocate nodes on the propeller palm base (as generated by PVFIL) so as to achieve bolt attachment points, and to identify nodes on the propeller palm base to which gap elements will be attached or boundary conditions applied. On the other hand should more accurate stresses be required in vicinity of bolt holes, detailed bolt and bolt structures need to be generated. PVBLTB allows the user to interactively relocate nodes through the thickness of the propeller palm model (from PVFIL) so as to achieve suitably shaped elements that groups of them could be removed and replaced by detailed bolt and bolt hole substructure pairs with mating nodes under the bolt head for optional implementation of gap elements. After elements for removal are identified using the graphics cursor, a reduced palm model is generated. On its base nodes can be identified at which gap elements will be attached or boundary conditions applied. PVBOLT generates models of bolt and bolt hole substructure pairs while PVBLTC permits base nodes on the bolt hole substructures to be identified for gap element and boundary condition purposes.

Programs PVHUB, PVPALM, PVBLD2, PVKEY, PVBLTA, PVBLTB, PVBLTC and PVBOLT generate boundary condition information which is passed on to program PVBCON. This program processes boundary condition information obtained from relevant sources and also provides the capability for interactive input and/or modification of boundary conditions. At present, the boundary conditions for the hand generated model (PVBLD3) must be input interactively. Similarly gap element data generated by PVBLTA, PVBLTB, PVBLTC and PVBOLT is passed to program PVBCON for consolidation and possible user interactive modification.

PVCHL1 generates a detailed air channel model from a blade modelled with 20-noded solid elements. From a previous analysis of the blade only model, loading and boundary condition information are extracted to generate a PREFX.CHL file for the detailed analysis of the air channel model.

PVUSE generates the PREFX.USE file and submit control file PREFX.CTL for analysis.

6.3 Hydrodynamic Added Mass Finite Element Model

Programs PVFLU1 and PVFLU2 generate the data describing a finite element model of the fluid surrounding a propeller blade. At present, only the 20-noded fluid element and the corresponding 8-noded interface element are used.

The program requires, as input, the propeller radii and section data (i.e. fraction radius, pitch, chord, skew, and rake) and a description of the structural finite element grid, which is used to define the interface grid. This information must be provided by disk data files generated by PVBLD1 and PVBLD2.

A typical finite element mesh for the blade-fluid interface is shown in Figure 6.15. The solid lines in Figure 6.15 define the structural finite element grid; the dotted lines represent the fluid grid.

The structural finite element grid defines the fluid-structure interface. The user has the option to use all the nodes of the structural finite element grid or only a portion of them. By using only a portion of the structural nodes, the user is able to reduce the size of the problem. Also, for substructure analysis, this option can be used to generate added mass for the master nodes. However, when this option is used, the nodes selected must describe 8-noded surface elements, as shown in Figure 6.16. The corner and mid-side nodes are identified by specifying radial and clockwise positions selected from radial and chordwise positions used to define the original blade elements.

The distances of the leading and trailing edge fluid boundaries from the blade must be defined by the user. For highly skewed blades when fold over of fluid elements is likely to occur at the trailing edge, the program issues a warning if the maximum distance to avoid fold-over has been exceeded. PVFLU2 calculates the global co-ordinates for all corner nodes of the mid-surface. Sometimes, especially for highly skewed blades, it is necessary to adjust the boundaries of the fluid model. For this reason, the program PVFLU1 generates, initially, a thin layered model which is easy to visualize. Adjustments can be made interactively. In order to generate co-ordinates for the other sections, first a vector at mid-span normal to the mid-surface is defined, then, using specified layer depths, co-ordinates for the remaining corner nodes are computed. A maximum of 10 layers can be specified, i.e. five on the face side and five on the back side. All intermediate nodes (except those at the fluid-structure interface) are then computed using straight line interpolations.

PVFLU2 is supported by the plotting programs PGFLUD and PLOTHD, which can be used to plot the fluid finite element model.

Program PVFLU3 generates data for describing the surrounding fluid by the surface panel method. The program uses the finite element grid information generated by program PVBLD2.

Note: Fluid models cannot presently be generated with the optimized blade model (see Figure 6.6).

6.4 Load Generation

Three pre-processor programs are available for generating loads - PVLODA, PVLODB and PVLODV.

VAST incorporates a quadratic interpolation scheme to generate pressure distributions over thick/thin shell and 20-noded solid elements. Consequently, pressure spikes along the leading and trailing edges can lead to highly distorted pressure distributions. The program PVLODA is available for eliminating this problem. A smoothing technique is used, which produces the same thrust and spindle torque as the original pressure distribution. PVLODA is supported by the plotting program PGLOD1, which can be used to plot chordwise pressure distributions as well as time history plots for dynamic pressure.

PVLODB reads from cards, or data files PREFX.LIN, LAD or T13, a description of the blade pressure loading, expressed in terms of face and back normal pressures specified at radial and chordwise positions and lumped drag forces specified at radial positions. PGLOD2 can be used to plot the input, adjusted and chordwise finite element generated

For a static analysis, a single set of pressure loadings at face and back radial and chordwise positions are required; for a dynamic response analysis, pressure loadings at specified points (≥2) in time are required. Complete load time histories or one cycle of a repeated dynamic pressure loading must be supplied, with the option in PVLODV to automatically generate repeated load cycles. Practically any shape of pressure curve can be described. For instance, different starting times can be specified by supplying zero pressure up until the time when pressure is applied, and discontinuous or step pressure curves can be described by specifying two points at the same time value. Examples of typical pressure loading specifications are shown in Figure 6.17. It is important to note that each specified point in time is common to all the radial and chordwise positions at which the normal pressures are applied at anytime in the time history.

PVLODV uses the data from PVLODB and angular velocities to generate data file PREFX.LOD, which must be supplied to program VAST.

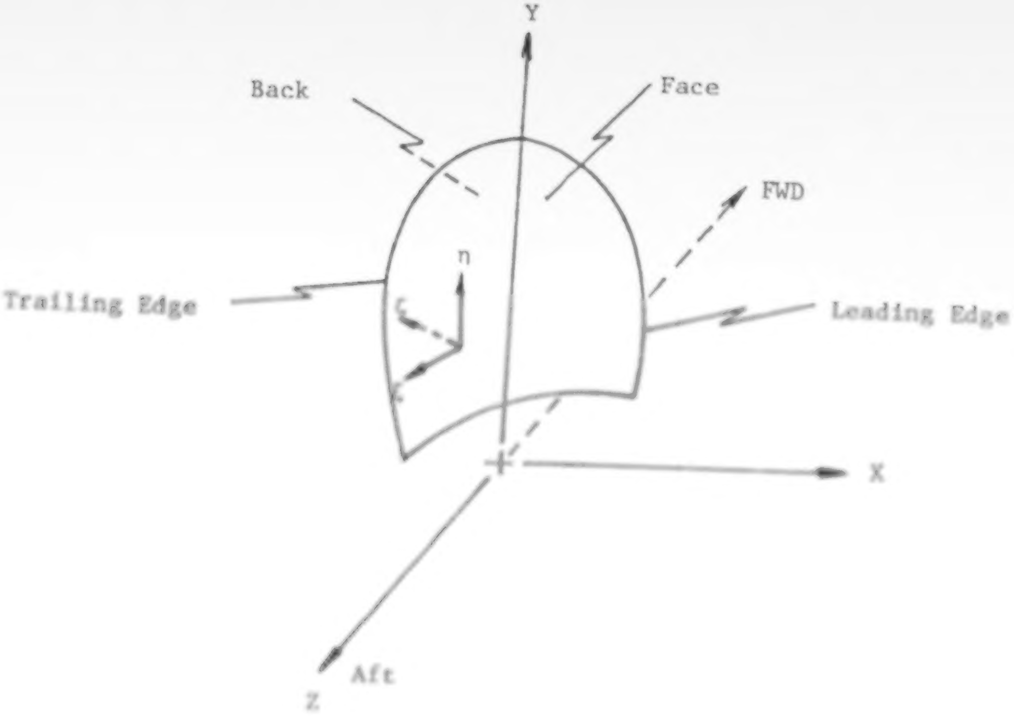

Y
Back
Face
FWD
Trailing Edge
Leading Edge
η
ζ
ξ
X
Z
Aft

FIGURE 6.1: Propeller Coordinate System

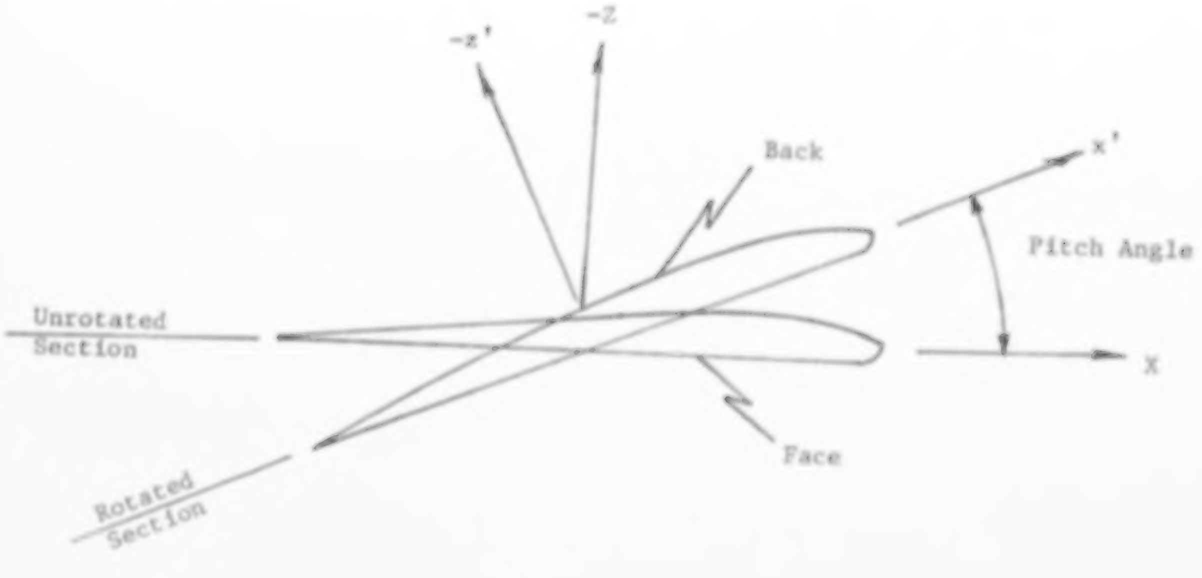
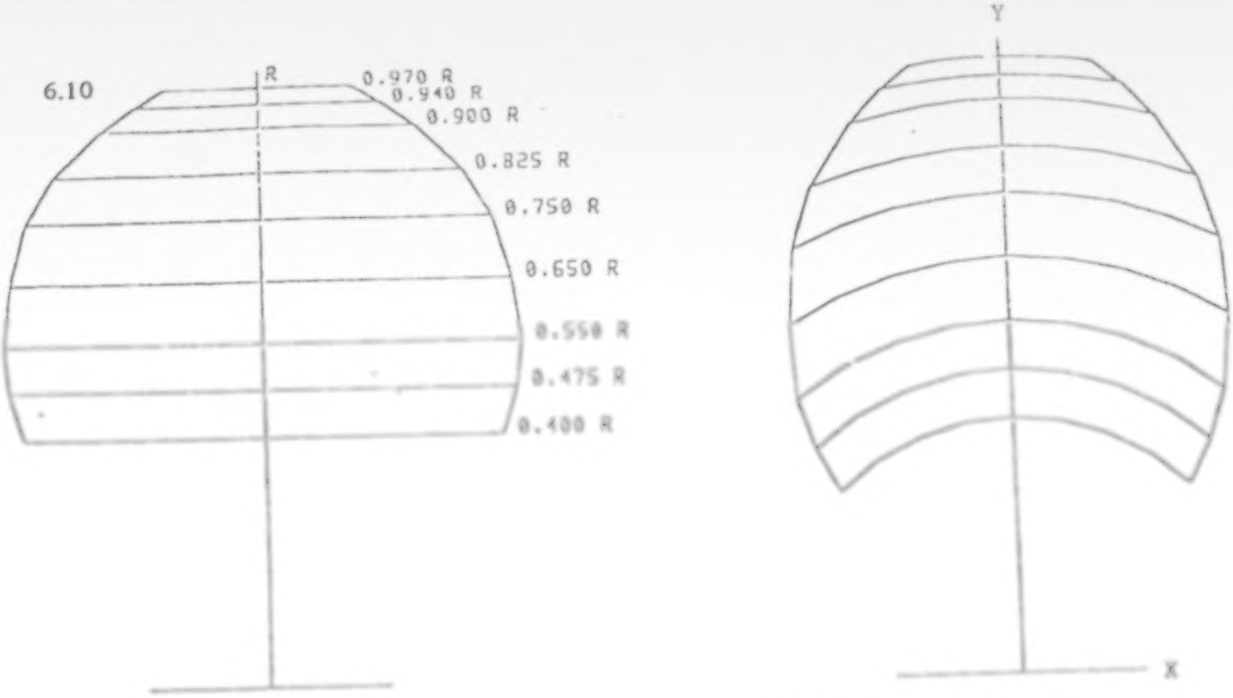
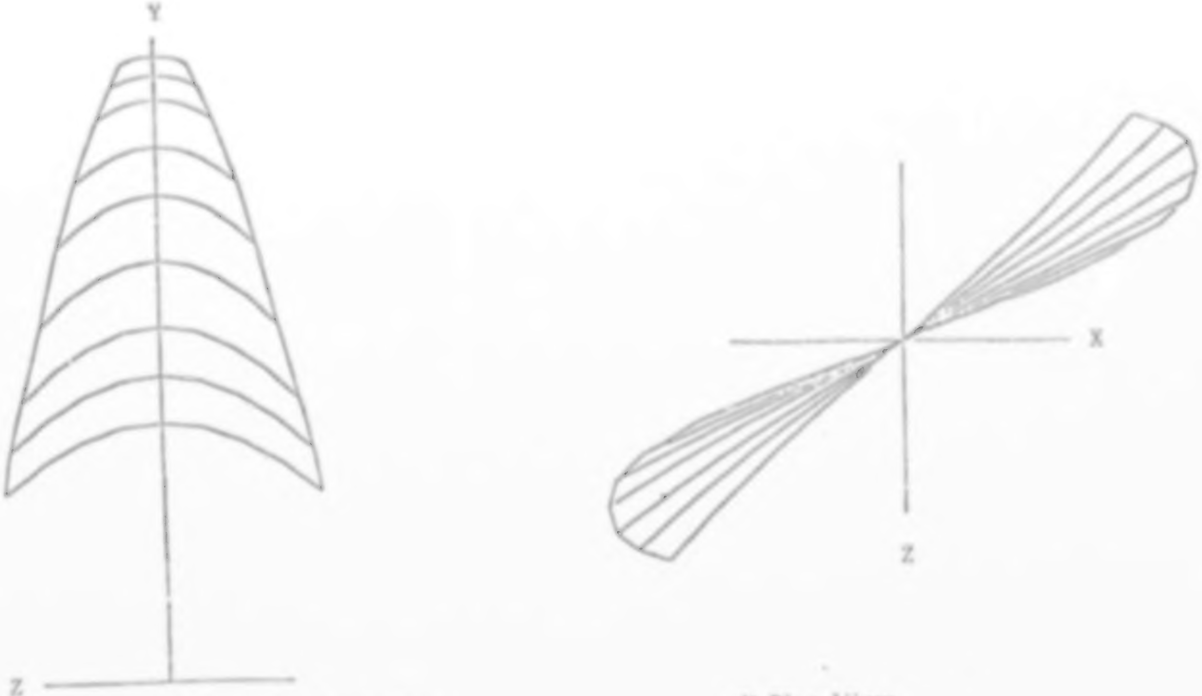

-z'
-Z
Back
x'
Pitch Angle
Unrotated Section
X
Face
Rotated Section

FIGURE 6.2: Developed Section


6.10
R
0.970 R
0.940 R
0.900 R
0.825 R
0.750 R
0.650 R
0.550 R
0.475 R
0.400 R
Y
X

a) Expanded Outline

b) Transverse Projection



c) Longitudinal Projection

d) Plan View

FIGURE 6.3: Various Outlines for a Typical Propeller Blade

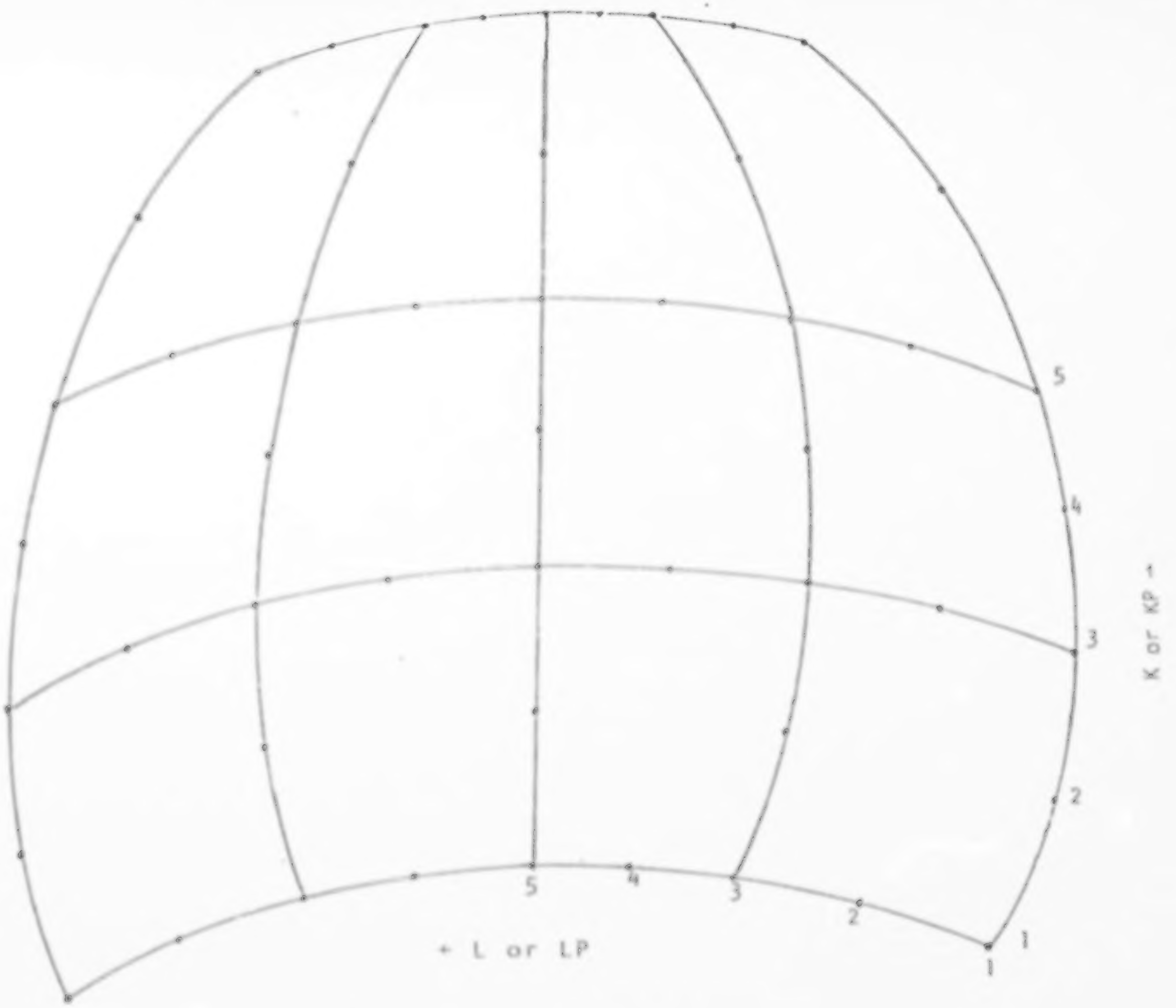
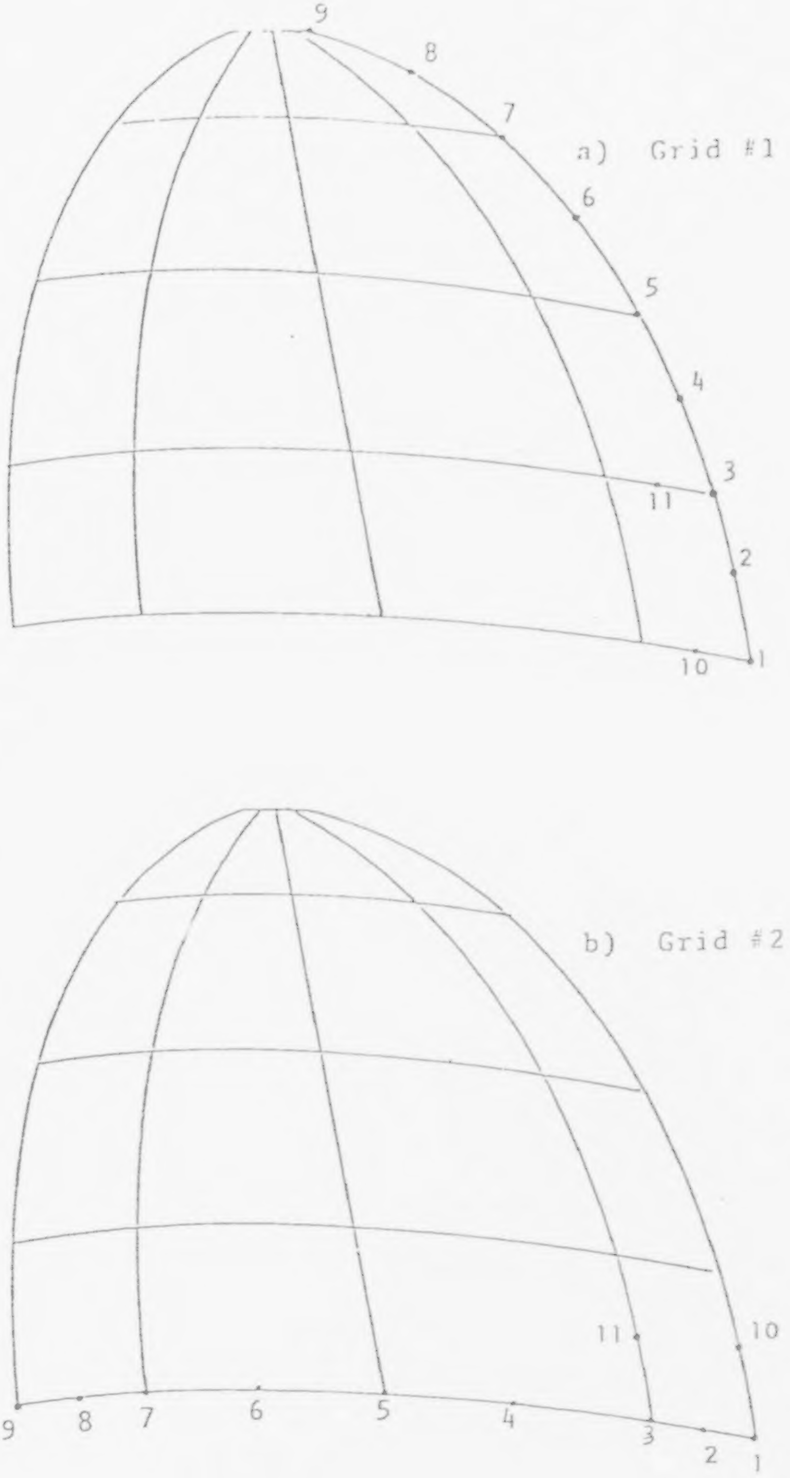

5
4
3
2
1
K or KP →
5
4
3
2
1
1
← L or LP

FIGURE 6.4: Use of Radial and Chordwise Positions for Locating and Nodes of the Finite Element Model


9
8
7
6
5
4
3
11
2
10
1
9
8
7
6
5
4
3
2
1
11
10

a) Grid #1

b) Grid #2

FIGURE 6.5: Node Numbering Scheme for Blade Model

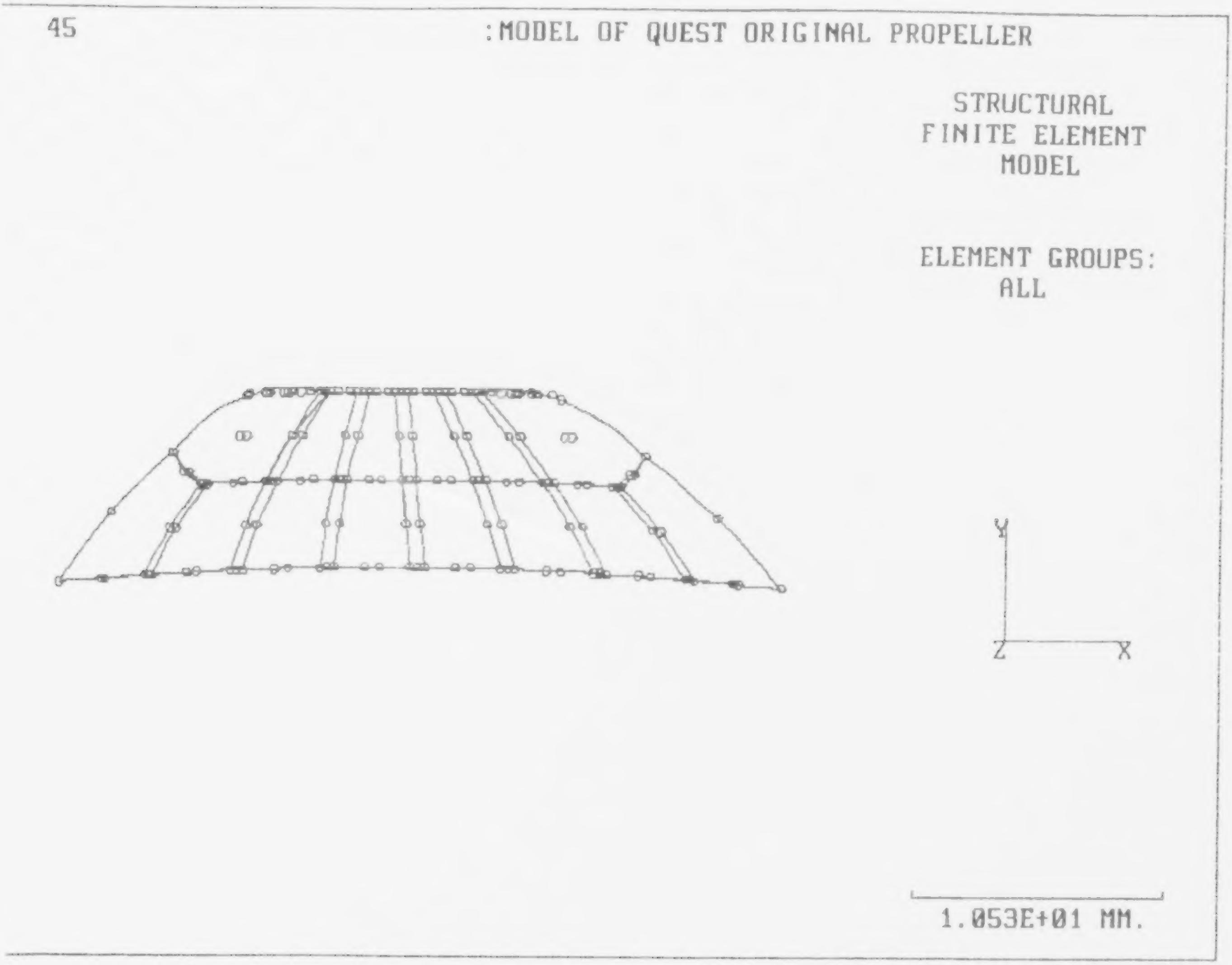
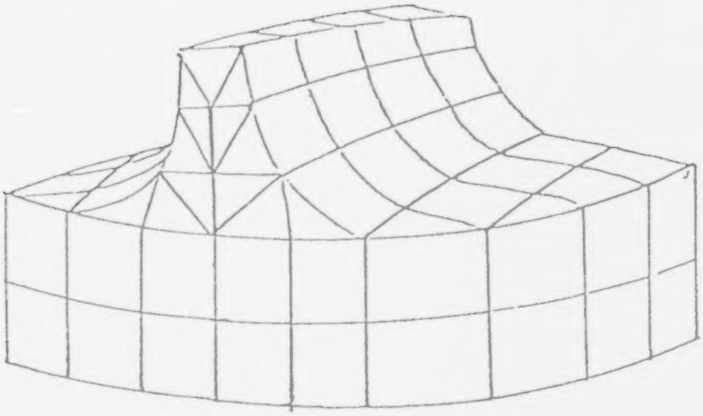
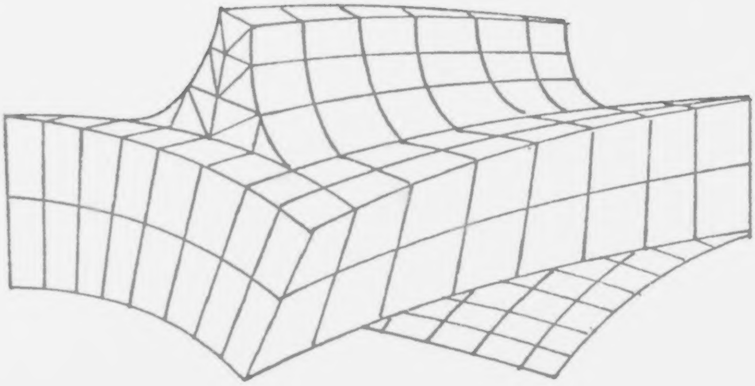

45
:MODEL OF QUEST ORIGINAL PROPELLER
STRUCTURAL
FINITE ELEMENT
MODEL
ELEMENT GROUPS:
ALL
Y
Z
X
1.053E+01 MM.

FIGURE 6.6: Optimized Mesh for Blade Tip (LOPTM=3)



(a) Palm



(b) Hub

FIGURE 6.7: Palm and Hub Finite Element Models

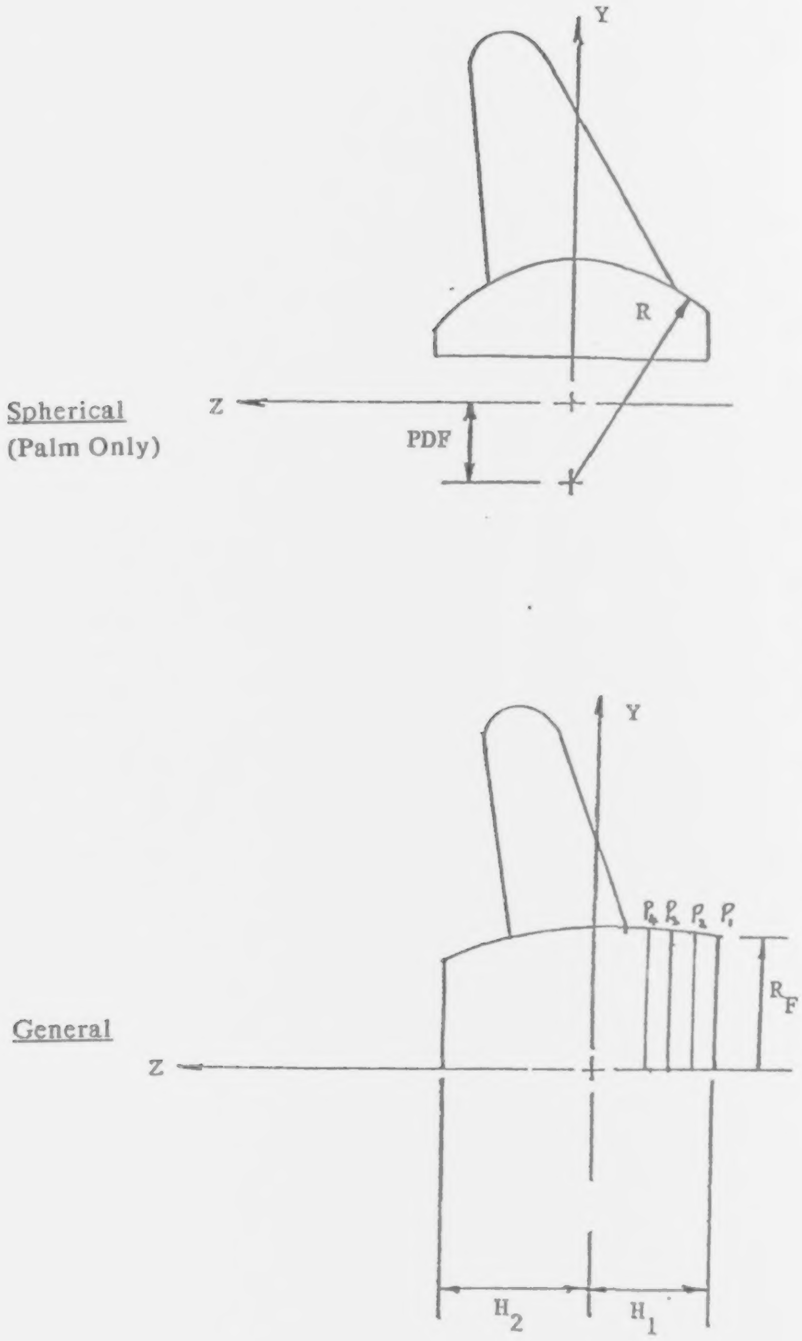

Y
R
Spherical
(Palm Only)
Z
PDF
Y
P4 P3 P2 P1
RF
General
Z
H2
H1

FIGURE 6.8: Types of Blade Palm/Hub Models

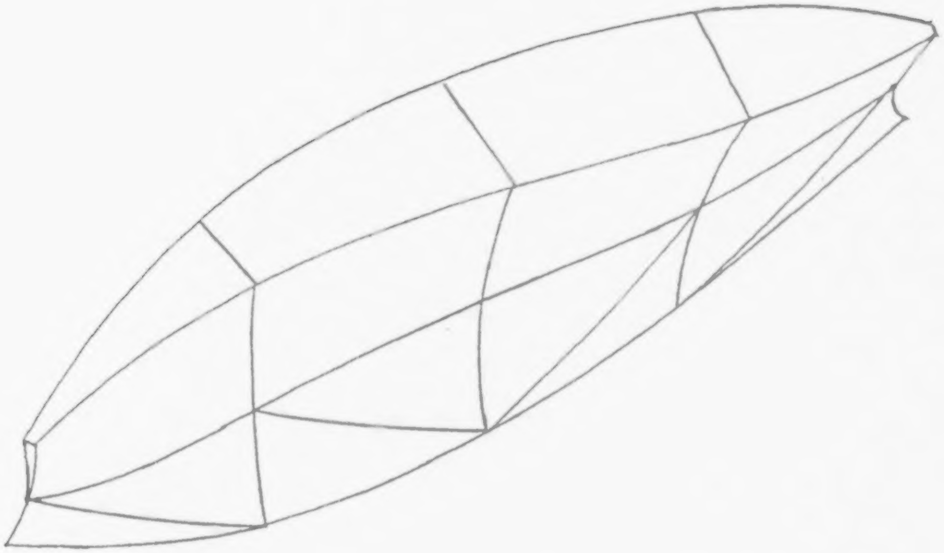


FIGURE 6.9: Typical Fillet Model

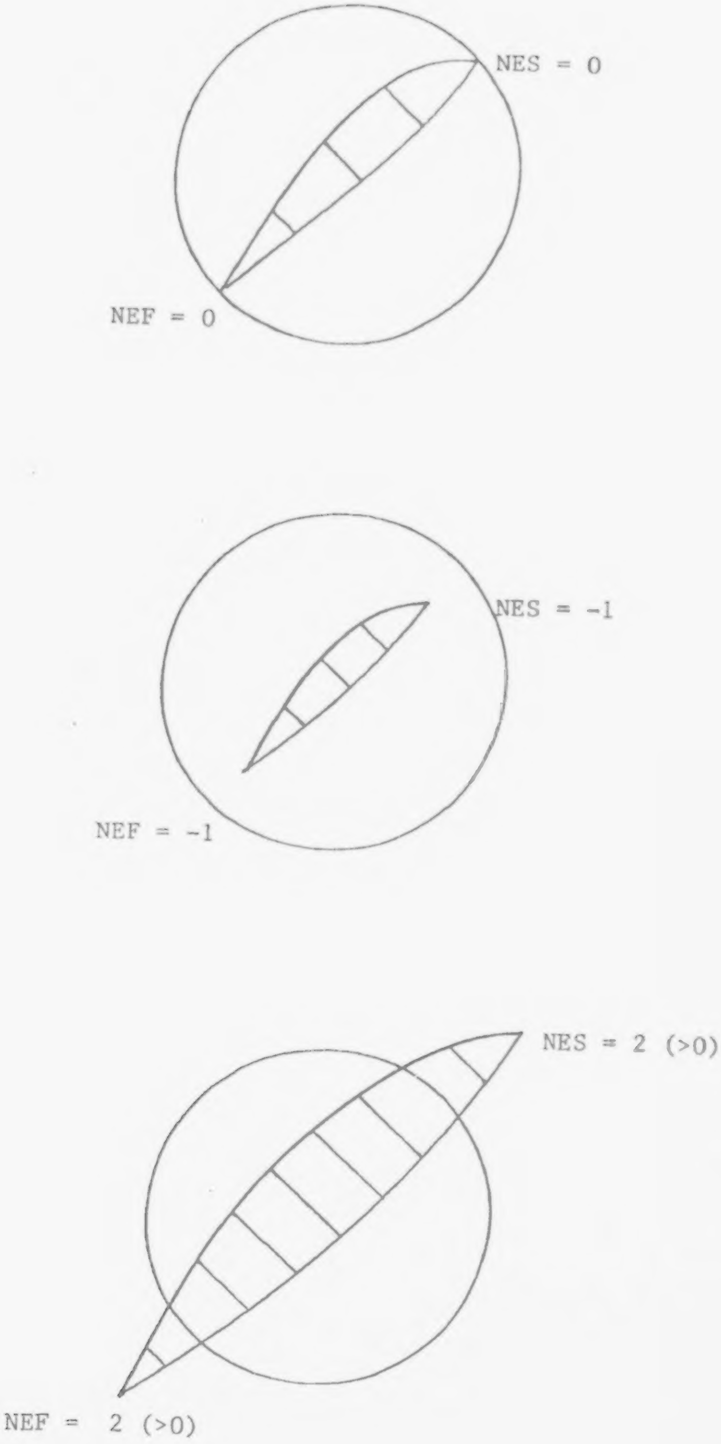
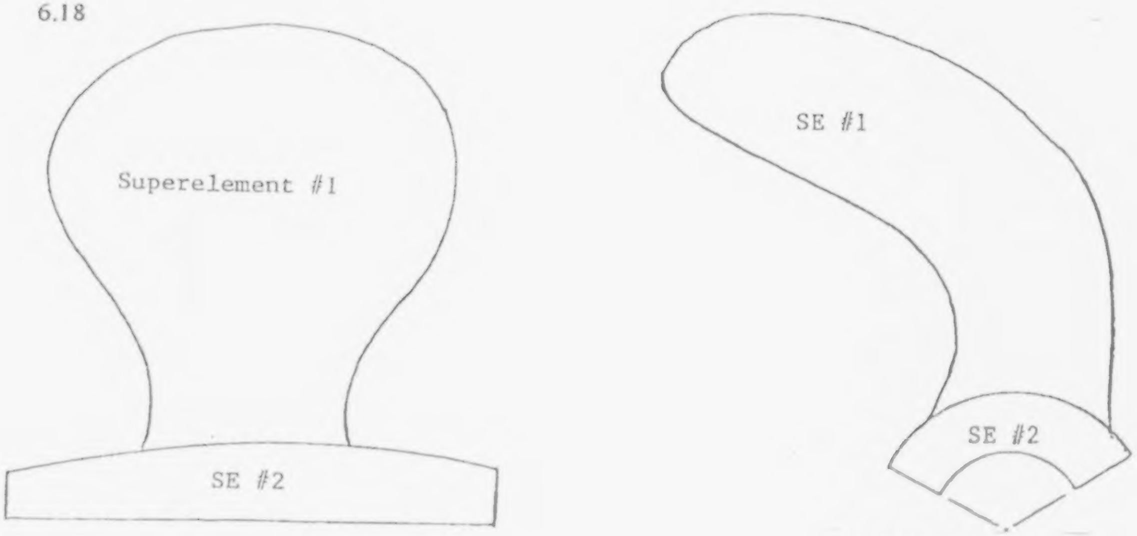

NES = 0
NEF = 0
NES = -1
NEF = -1
NES = 2 (>0)
NEF = 2 (>0)

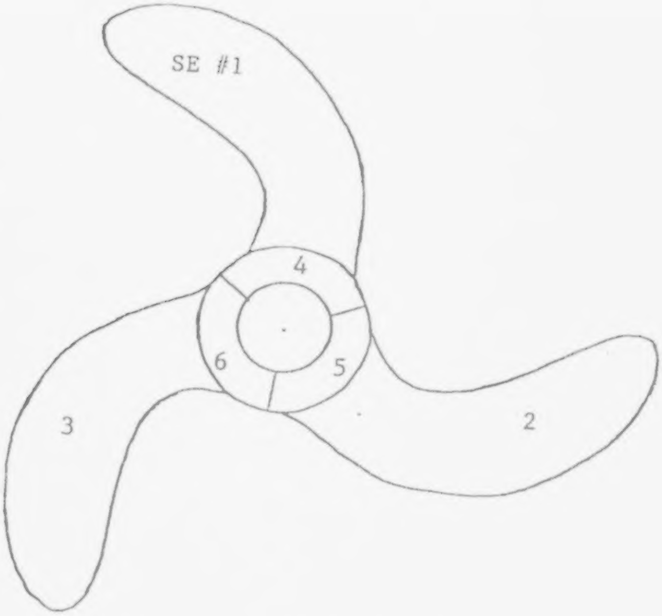
FIGURE 6.10: Location of Root Base End Relative to Leading and Trailing Edges of the Blade


Superelement #1
SE #2
SE #1
SE #2

Palm Model

Hub Model

(a) Single Blade Model



(b) Complete Blade-Hub Model

FIGURE 6.11: Types of Propeller Models

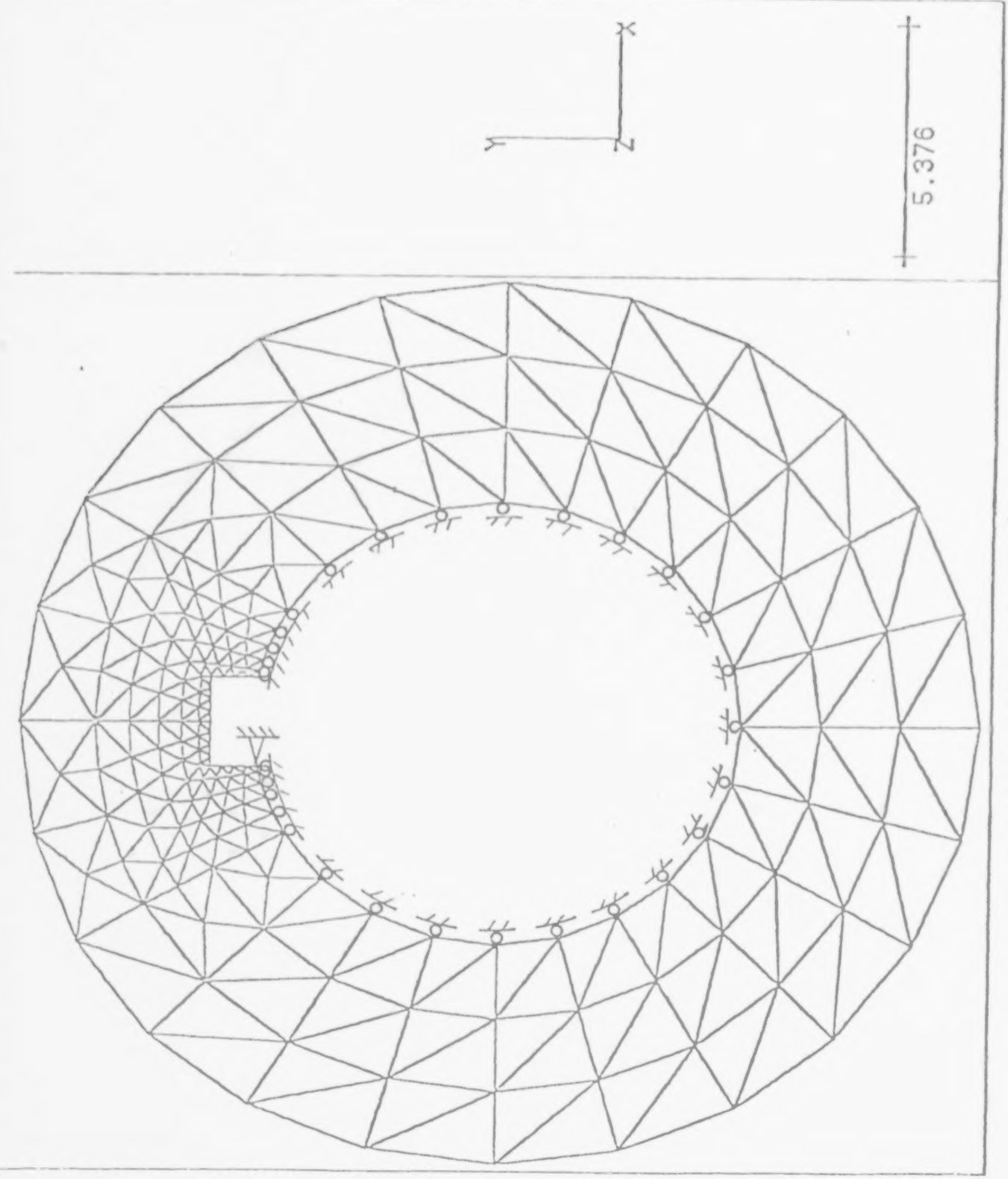


FIGURE 6.12: Finite Element Model of Keyway

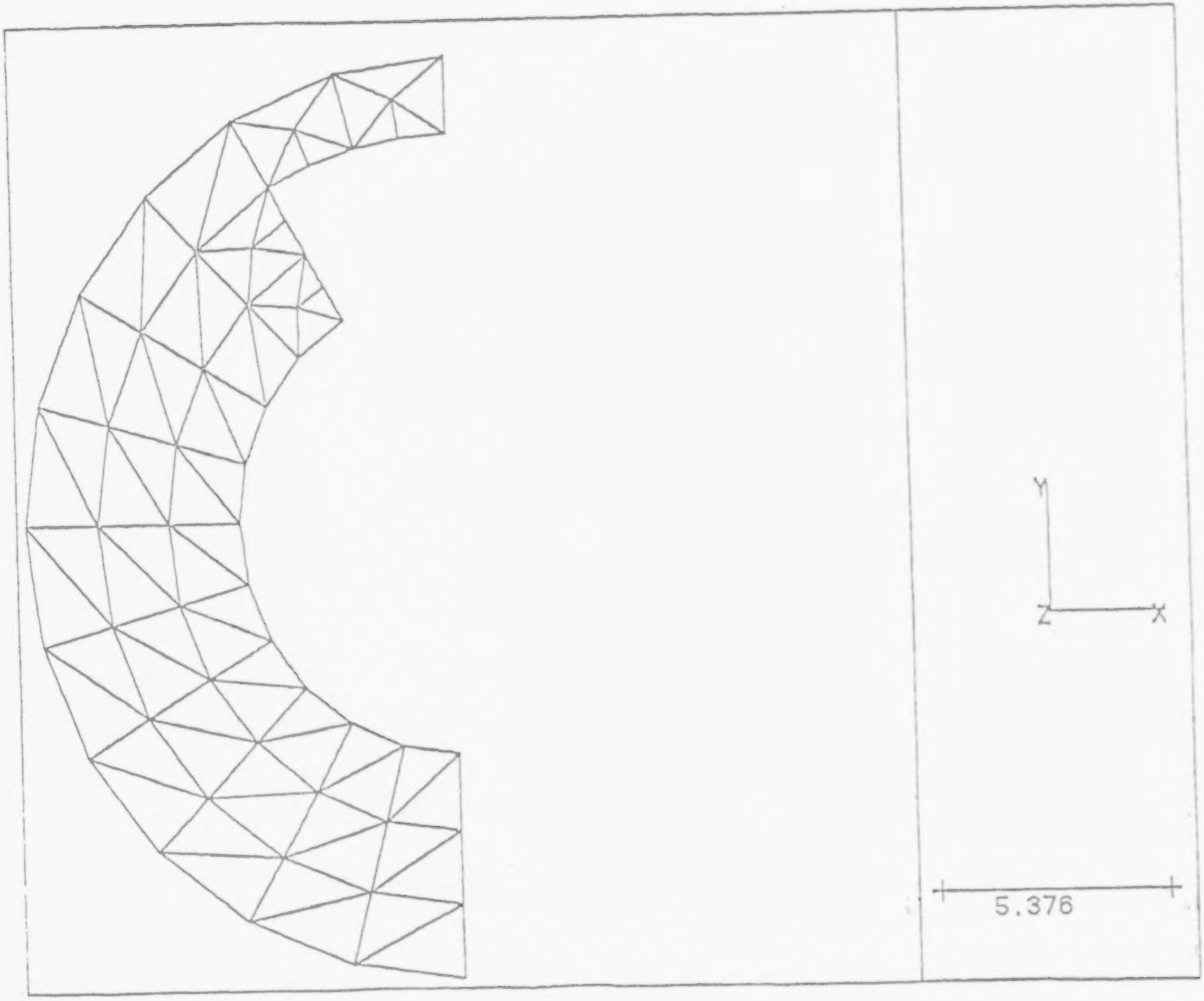


FIGURE 6.13: Keyway Model - Superelement #1

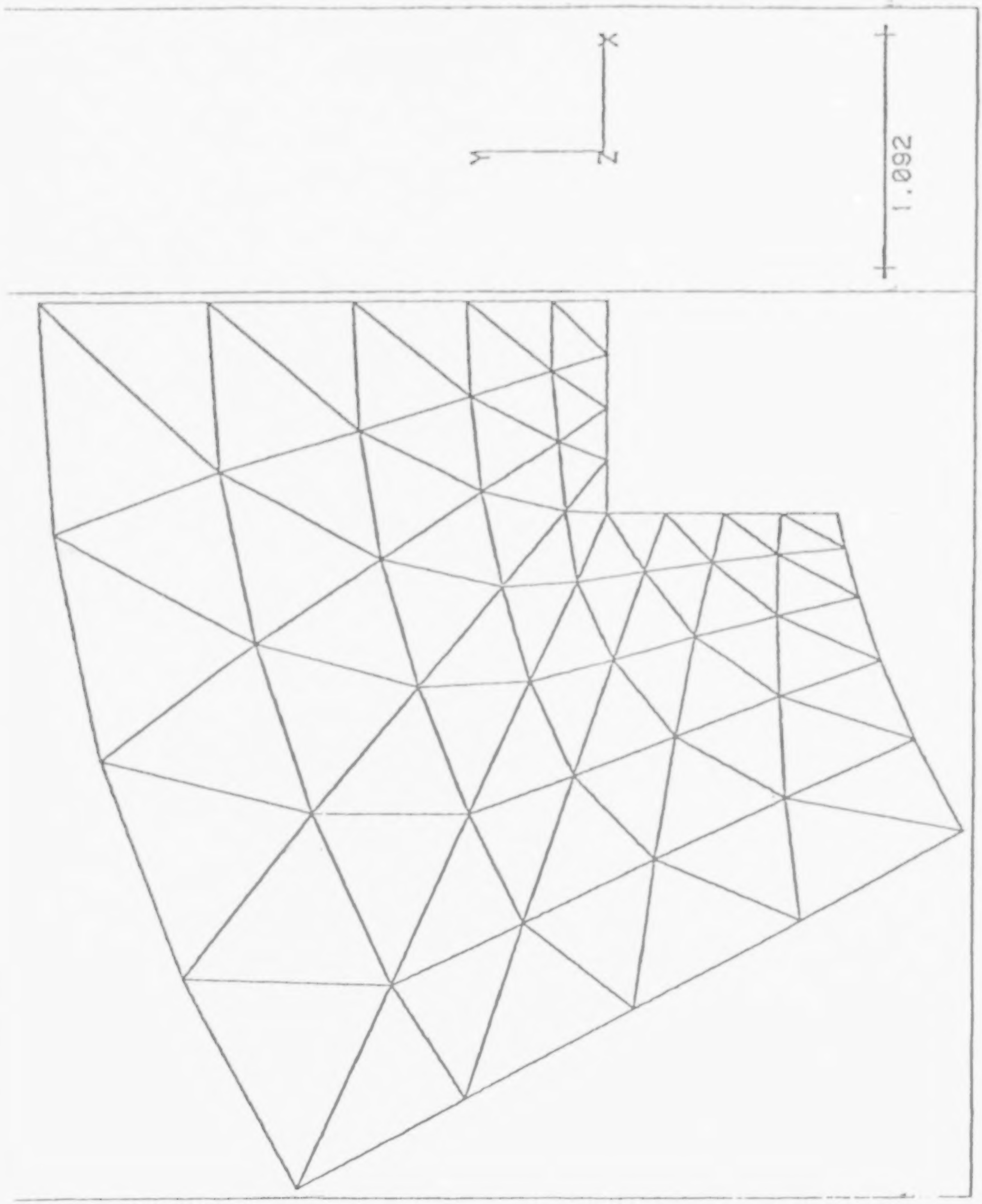


FIGURE 6.14: Keyway Model - Superelement #3

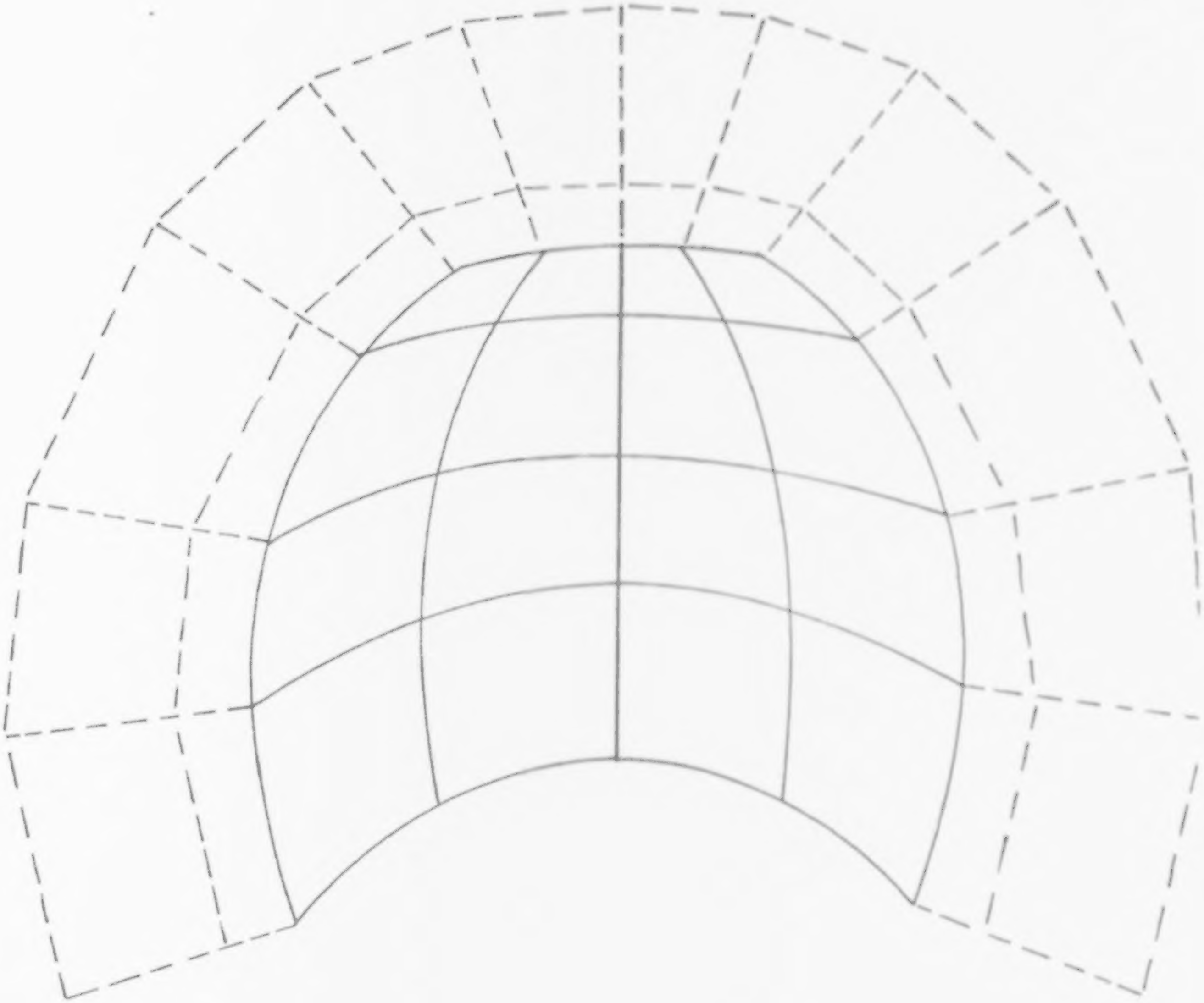


FIGURE 6.15: Finite Element Mesh for a Typical Propeller-Fluid Model

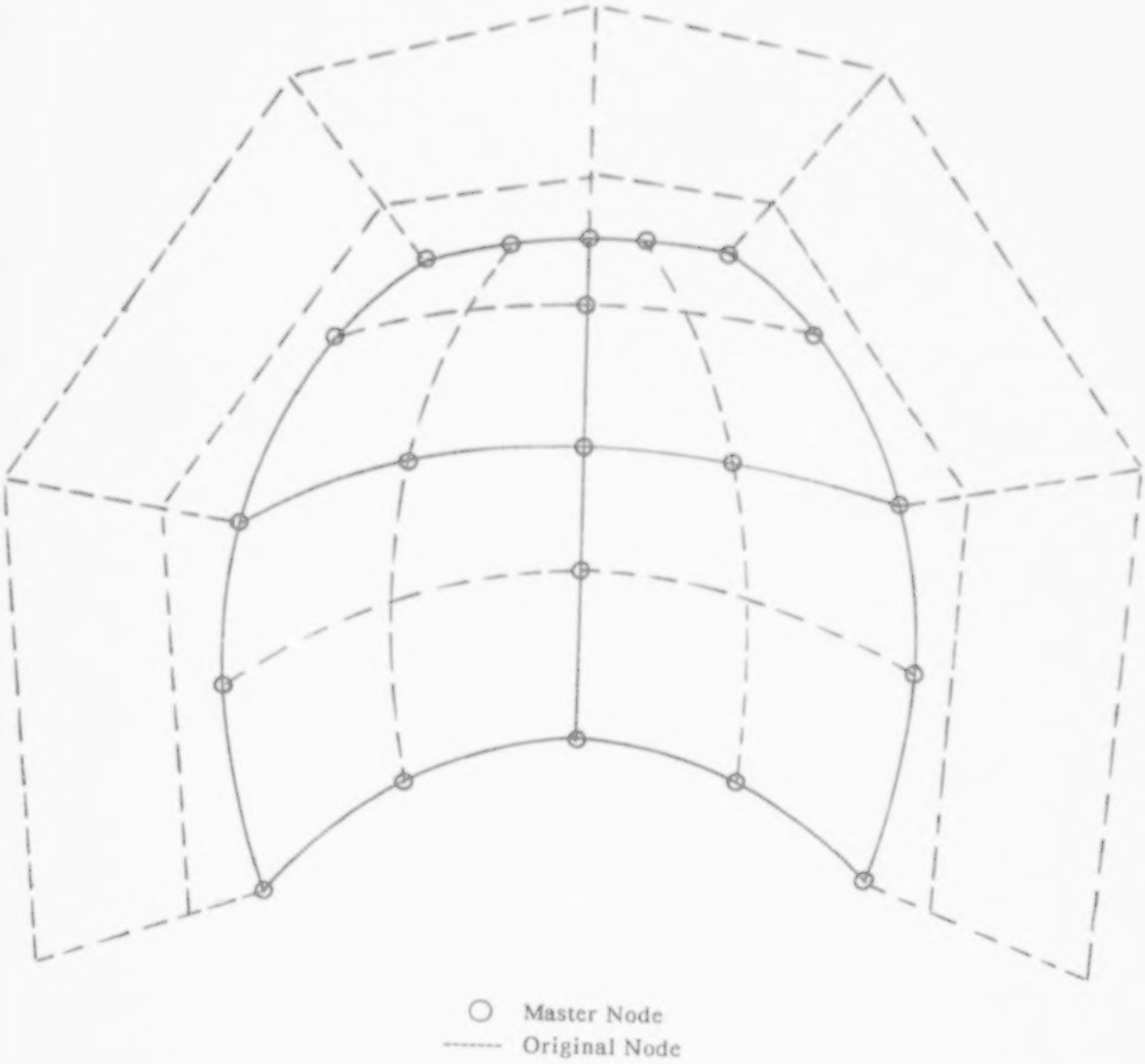
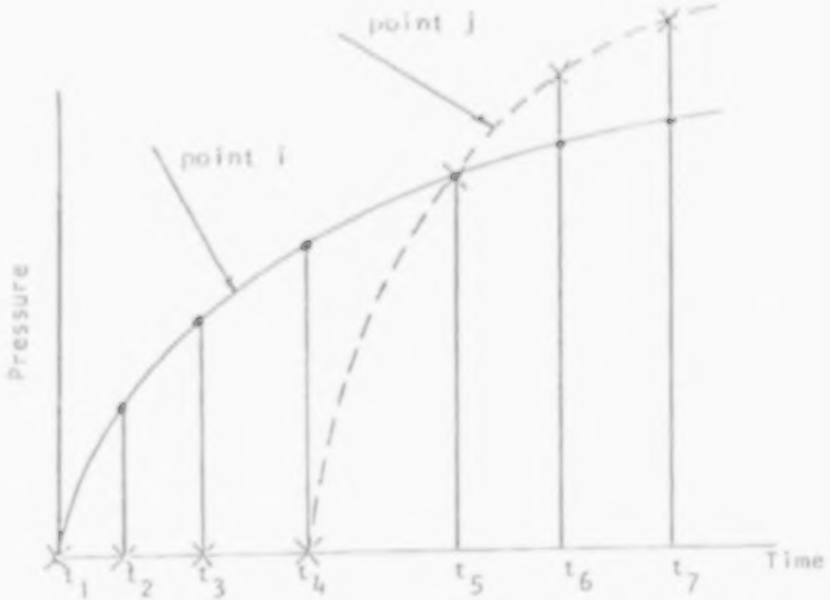
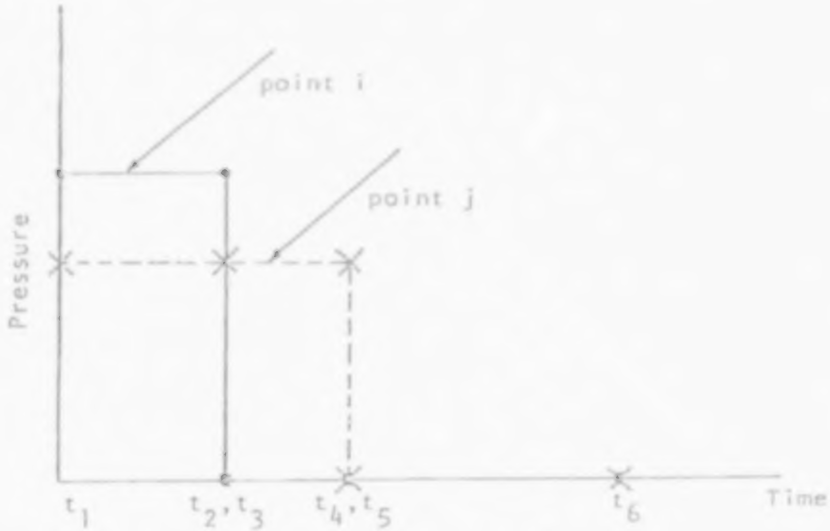

○ Master Node
------- Original Node

FIGURE 6.16: Reduced Finite Element Mesh for a Typical Propeller-Fluid Model


point j
point i
Pressure
t_1
t_2
t_3
t_4
t_5
t_6
t_7
Time

(a) Different Starting Times


point i
point j
Pressure
t_1
t_2, t_3
t_4, t_5
t_6
Time

(b) Discontinuous Pressure Curves

FIGURE 6.17: Examples of Typical Pressure Time Histories

7.0 INTERPRETATION AND PRESENTATION OF RESULTS USING POST-PROCESSING PROGRAMS

The interpretation and presentation of results involves the use of the post-processing programs described in Tables 5.2 and 5.4. A flow diagram showing the order in which these programs must be executed is shown in Figure 5.2.

8.0 GUIDELINES ON HOW TO USE PVAST EFFECTIVELY AND EFFICIENTLY

Following are suggestions on how to use PVAST effectively and efficiently:

(i) Model Selection

If a stress or a frequency analysis of the blade only is required, use the blade model only option. If the fillet radii are small compared to the blade thickness at the root, and the blade to palm/hub intersection coincides reasonably well with a section of constant radius on the blade, the option to incorporate a portion of the root fillet need not be used. This avoids generating a root model. If stresses in the fillet are required, then the palm/hub segment must be modelled. In the case of a palm, if a detailed stress analysis of the bolt holes is not required, the option to model bolt holes and bolts (using constrained substructures) should not be used. Also, if bolt loads are not required, the use of rod and gap elements should be avoided. This will reduce the problem size and eliminate the iterative procedure for computing gap element forces. Very rarely should a completely assembled model of the entire propeller be used, as such an analysis will require a vast amount of computer time and disk storage. Therefore, whenever possible, the results from a single blade analysis should be used for shaft related analyses.

(ii) Mesh Selection

In general, the fineness of the blade mesh for a stress analysis should be governed by the load distribution, stress gradients expected, and accuracy desired. For a vibration (natural frequency) analysis the governing factor is the accuracy and number of modes desired. Even for relatively fine grids computer costs for a static stress analysis of the blade alone will not be excessive. However, for dynamic problems (i.e. dynamic response and natural frequency analysis) where the added mass of the surrounding water is to be considered, fine blade meshes can lead to large computer costs. These factors should be considered when selecting the mesh for a blade only analysis.

At present, in PVAST, the fineness of the palm/hub mesh will be dictated by the fineness of the blade mesh. Consequently, a mesh required to do a detailed and accurate blade analysis will result in a palm/hub model with many degree-of-freedoms, and computer costs will be large. In such cases it may be advisable to use two different blade models, i.e. one of a fine mesh for a blade only analysis, and one of a coarse mesh, to be used in the blade - palm/hub analysis. This two blade model approach can also be used in blade stress and natural frequency analyses, where the fine mesh is used for computing stresses and the coarse mesh is used for computing natural frequencies.

(iii) Solutions Schemes

The user has to select one of two methods for natural frequency analysis, and one of two methods for time history (dynamic) analysis. Guidelines for selecting the appropriate schemes are given in Sections 2.3 and 2.4.

(iv) Error Detection

This is a very important consideration in any finite element analysis. It involves verification of the models (i.e. geometry, material properties, etc.) and diagnostic checks (encountered during the solution phase).

The computer graphics facilities available in PVAST (described in this manual) should be fully utilized to verify geometry, and if applicable, loading. Data such as Young's Modulus, material density, etc., which cannot be checked graphically, should be checked by looking at computer printouts.

VAST (which handles the solution phase) has a number of built-in error monitoring facilities. They range from model integrity tests to numerical error estimations. Model integrity tests include checks at the input data stage (i.e. distortion, angle and aspect ratio) and at the assembled stiffness matrix stage (i.e. disparate stiffness, boundary conditions and boundary spring magnitudes). Model integrity tests performed on input data identify potential error sources and enable corrections to be made without expending large amounts of computational effort needlessly. Numerical error estimations are intended for the advanced user with a knowledge of numerical analysis. Equation conditioning indicators based on pivot ratios (obtained during matrix decomposition) and force residuals (obtained after solution) are provided. VAST also offers a procedure for calculating condition numbers. However, this facility requires an eigenvalue analysis and, therefore, can be quite costly. The method is described in Reference 7.

9.0 REFERENCES

1. M.E. NORWOOD. "A Digital Computer Program for the Vibration and Stress Analysis of Propellers - User's Manual", Canplan Oceanology Limited, February 1975.

2. M.E. NORWOOD. "Application of the Finite Element Method for the Calculation of the Hydrodynamic Added Mass of Marine Propeller Blades", Martec Limited, Halifax, 1977.

3. S. AHMAD, B.M. IRONS, and O.C. ZIENKIEWICZ. "Analysis of Thick and Thin Shell Structures by Curved Finite Elements", Int. Journal for Numerical Methods in Engineering, Vol. 2, 419-451, 1970.

4. M.E. NORWOOD. "Analysis of Shell Structures by Curved Finite Elements", Canplan Oceanology Limited Technical Report, Halifax, July 1974.

5. M.E. NORWOOD. "Vibration and Stress Analysis of Marine Propellers by Finite Element Analysis - Part II", Canplan Oceanology Limited, Halifax, 1976.

6. M.E. NORWOOD. "Propeller Vibration and Stress Analysis by Finite Element Methods (PVAST): User's Manual (1980), Martec Limited, Halifax, 1980.

7. "Vibration and Strength Analysis Program (VAST): User's Manual Version #06, Martec Limited, Halifax, Nova Scotia, 1990.

8. M.E. NORWOOD. "Vibration and Stress Analysis of Marine Propellers by Finite Element Methods: Part IV.

9. J.S. PRZEMIENIECKI. Theory of Matrix Structural Analysis, McGraw-Hill, 1968.

10. MOVIE.BYU: A General Purpose Computer Graphics System, Version 3.6, Brigham Young University, January 1980.

11. O.C. ZIENKIEWICZ. "The Finite Element Method" (Third Ed.) McGraw-Hill Book Company (UK) Ltd., 1977.

12. T.A. VERNON, B. BARA, and D. HALLY. "A Surface Panel Method for the Calculation of Added Mass Matrices for Finite Element Models", Technical Memorandum 88/203, DREA, 1988.

13. S.J. CARLTON. "Marine Propeller Blade Stresses", Transactions of the Institute of Marine Engineers, February 1984, ISSN 0309-3948 Vol. 96, 1984, Paper 63.

14. D.R. SMITH and T.R. MacFARLANE. BLADE: A Propeller Blade Geometry Generator Program : User's Manual. DREA Technical Communication 91/302, February 1991.

APPENDIX A

INPUT DATA FOR PROGRAM VAST

The input data requirements for VAST are described in Reference 7. The main parts of the data consist of:

(i) Master control codes;

(ii) Nodal point coordinates, material properties and element connectivities;

(iii) Superelement data if substructuring is used;

(iv) Boundary conditions; and

(v) Load data.

The bulk of the data is found in (ii), (iii) and (v) and can be supplied to VAST via special files (eg. PREFX.T10, PREFX.T17, and PREFX.T27 generated by PVFIL/HUB/PALM, PVBLD2, PVBLD3, and PVSUB1/2; and PREFX.LOD generated by PVLODV).

APPENDIX B

INPUT DATA FOR GENERATING BLADE AND ROOT MODELS

B.1 PVBLD1: Computation of the true space coordinates for propeller blades using the designer's basic blade design data.

Header Card (A6)

BLADE1 (see Appendix K for further details on use of header cards)

Card 1 (2I5)

IDATA = Flag used to specify blade data sources:
1, read CARDS 2-10 from PREFX.CDR
2, read CARDS 2-10 from PREFX.T16 (formatted).

IPTC = 0, printout (of blade geometry and finite element grid) is reduced.

Card 2 (A72)

TITLE = 72 character title.

Card 3 (I5)

ICHORD = Code for specifying chordwise positions for defining face and back offsets. If =1, chordwise positions for the first section are provided, and those for the remaining sections are assumed to be identical. If =2, chordwise positions are provided for each section.

Card 4 (2I5)

K = Number of radial positions where data are provided. ($3 \leq K \leq 43$)

L = Number of chordwise positions where data are provided. ($3 \leq L \leq 41$)

Card 5 (F10.4)

RR = Propeller full radius (length).

Card 6 (See Figure B.1)(5E10.3)

R = Section radial location as fraction of full radius.

P = Pitch angle (degrees), positive counter clockwise.

S = Section chord length (length).

SK = Skew (length), positive for section shifted in direction of trailing edge.

RK = Rake (length), positive aft.

Card 7 (C_{1j}, j=1,L) (8F10.4)

$C_{(1,1)}$ ↓ $C_{(1,L)}$ = The chord fractions for the face and back surface offsets for the first section (see Figure B.1).

Repeat Card 8 for i=2,K

Card 8 (C_{ij}, j=1,L) (Omit if ICHORD=1) (8F10.4)

$C_{i,1}$ ↓ $C_{i,L}$ = The chord fractions for the face and back surface offsets for sections 2 to K.

Repeat Cards 9 and 10 for j=1,L

Card 9 (TB_{ji}, i=1,K) (8F10.4)

$TB_{(j,1)}$ ↓ $TB_{(j,K)}$ = Back surface offsets (length), i.e. the z' coordinate in the developed section (see Figure B.1). (Negative value for -z' coordinate.)

Card 10 (TF_{ji}, i=1,K) (8F10.4)

$TF_{(j,1)}$ = Face surface offsets.
↓
$TF_{(j,K)}$

Card 11 (2I5)

NEK = Number of finite element grid divisions in radial direction (NEK$\leq$21).

IRPC = 0, grid to be proportioned by user.
= 1, grid to be automatically biased towards root and tip (even proportioning in radial direction is a special case).
= 2, grid to be automatically biased towards root only.
= 3, grid to be automatically biased towards tip only.

Card 12(a) (I5) (Omit if IRPC $\neq$ 0)

NRLN = Number of chordwise positions at which fraction radii are to be provided.
= 1, fraction radii provided at leading edge alone.
= 2, fraction radii provided at leading edge (first) and trailing edge (second).
= 3, fraction radii provided at leading edge (first), trailing edge (second) and mid-chord (third).

Card 12(b) (8F10.4)

$RP_{(1)}$ = The fraction used to define the finite element grid divisions in the radial direction, provided at 1 to NRLN chordwise positions.
↓
$RP_{(NEK+1)}$

NOTE: Fraction radii at each chordwise position begins on a new card.

Card 13 (3F10.4) (Omit if IRPC = 0)

RF = First fraction radius for finite element grid.

RL = Last fraction radius for finite element grid.

GPFR = Geometric progression factor for biasing (1.0 if grid is to be evenly proportioned in radial direction).

Card 14 (3I5)

NEL = Number of finite element grid divisions in chordwise direction (Maximum of 20).

ICPC = 0, grid to be proportioned by user.
= 1, grid to be automatically biased towards leading and trailing edges (even proportioning in the chordwise direction as a special case).
= 2, grid to be automatically biased towards leading edge only.
= 3, grid to be automatically biased towards trailing edge only.

ICORDE = 0 (when ICPC $\neq$ 0)
= 1, chordwise positions are the same for all radial positions.
= 2, chordwise positions are to be specified for each radial position.

Card 15 (CP_{1j}, j=1, NEL+1) (8F10.4) (Omit if ICPC $\neq$ 0)

$CP_{(1,j)}$ = The fraction chord positions used to define the finite element grid
↓ divisions (for all radial positions if ICORDE
$CP_{(1,NEL+1)}$ = 1, or for the first radial position if ICORDE = 2).

Repeat Card 16 for i=2, NEK+1

Card 16 (CP_{ij}, j=1, NEL+1) (8F10.4) (Omit if ICPC $\neq$ 0 or ICORDE = 1)

$CP_{(i,1)}$ = The fraction chord position used to define to the finite element grid
↓ divisions for radial position i. Note: each radial position begins on
$CP_{(i,NEL+1)}$ a new card.

Card 17 (3F10.4) (Omit if ICPC = 0)

CF = First fraction chord for finite element grid.
CL = Last fraction chord for finite element grid.
GPFC = Geometric progressing factor for biasing (1.0 if grid is to be evenly proportioned in chordwise direction).

Card 18 (I5)

LOPTM = parameter for optimization of the finite element grid near the blade tip.
= 0 for no optimization.
= 1 for optimization at the leading edge.
= 2 for optimization at the trailing edge.
= 3 for optimization at both edges.

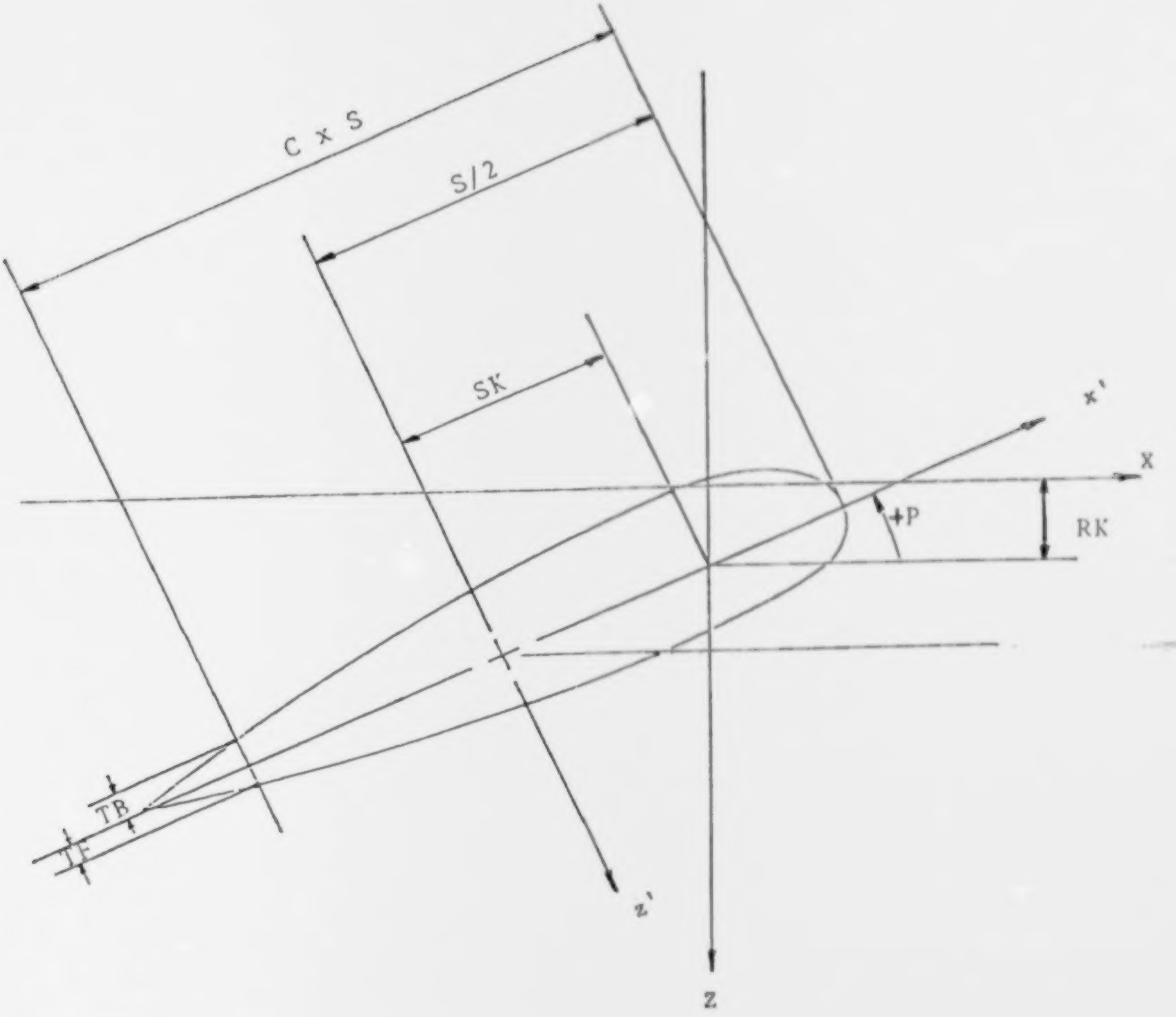

C x S
S/2
SK
x'
X
+P
RK
TB
TF
z'
Z

FIGURE B.1: Diagrammatic Explanation of Parameters Used to Describe Section Geometry

B.2 PVFIL, PVHUB, PVPALM: Generation of finite element models for fillet and propeller palm or hub.

Header Card (A6)

ROOT (See Appendix K for additional details on use of header cards)

Card 1 (7I5)

IOPT1 = Flag for defining type of root geometry (see Figure B.2).
1, palm model only.
2, general hub model.

IFR = 0, single radius fillet (same for face and back), where the radius is constant along chord.
1, single radius fillet (different for face and back), where the radius varies along chord.
2, two radii fillet (same for face and back) based on local blade thickness, as shown in Figure B.3, where the parameters AK1, AK2 and AK3 are constant along chord.
3, two radii fillet (different for face and back), as shown in Figure B.2, where the parameters AK1, AK2 and AK3 vary along chord.

ICF $\neq$ 0, continuous function fillet produced (valid only for IFR=2 or 3)

NCST = Number of chordwise stations at which fillet data is specified (valid only for IFR=1 or 3 also for IFR=2 when $ICF \neq 0$) ($NCST \leq 17$).

IPTC = 0, printout (of hub geometry and finite element data) reduced.

IOPT2 = 2, two elements along fillet edge [see Figure B.4(a)].
3, three elements along fillet edge [see Figure B.4(b)].

IOPT3 = 1, palm model.
2, hub model.

Card 2 (3E10.3)

E = Young's Modulus (force/length^2).

PR = Poisson's ratio.

DEN = Material weight per unit volume/g (force-time^2/length^4).

Card 3 (2I5)

NES = Number of elements in the blade leading edge overhang.

NEF = Number of elements in the blade trailing edge overhang.

Note: If NES = -1, palm or hub extension of one element layer at the blade leading edge. Similarly for NEF at the trailing edge (see Figure 6.9).

Card 4 Palm/hub geometric data. Provide Cards 4(a) or 4(b) for a palm or hub model, respectively.

Repeat Card 4(a) only if IOPT1=1

Card 4(a)(i) (E10.3)

R = spherical palm surface radius (length).
= 0 for general palm surface.

Card 4(a)(ii) (I5)

NP = Identifier for type of palm surface.
= 0 for spherical palm surface.
= >0 for general palm surface. In this case the program automatically sets NP to 16 irrespective of the value supplied. (See Figure B.5)

Card 4(a)(iii) (3E10.3) If NP>0 only.

XC, YC, ZC = X, Y, Z coordinates (length) of the points defining the patch for the palm surface. Provide this card once for each of the 16 points. Patch should be made larger than the palm surface. Points should be evenly spaced along 4 evenly spaced curves.

Repeat Card 4(b) only if IOPT1=2

Card 4(b)(i) (2E10.3)

H_1, H_2 = Distances from origin to front and aft ends, respectively (length). (Distances should be input as positive values.)

Card 4(b)(ii) (I5)

NP = Number of points defining hub shape.
NP should preferably be ≥4 since a cubic spline is used.

Card 4(b)(iii) (2E10.3)

ZC, YC = Z, Y (Radius) coordinates of points defining hub shape. Provide this card once for each of the NP points.

Card 4(b)(iv) (I5)

INTRP = parameter for type of interpolation to be used for splined curve defining the hub shape.
= 0 for normal interpolation (adequate for evenly spaced points).
= 1 for intrinsic interpolation (for very unevenly spaced points).

Card 5 (3E10.3) For IOPT3 = 1 only.

PBH = Palm base height (length).

PDF = Palm coordinate offset. PDF can be non-zero only for spherical palm model. Note that if PDF ≠ 0, then spherical radius R (Card 4) is taken from this offset and not the Z axis (see Figure B.2).

PD = Palm diameter (length).

Note: See Figure B.6 for definition of variables.

Card 6 (4E10.3, I5) For IOPT3 = 2 only.

RIDF = Internal hub radius at front end of hub (length).

RIDB = Internal hub radius at aft end of hub (length).

PHI = Included angle for bladed and bladeless hub segment pair (degrees).

PHIF = Included angle factor defining size of bladeless hub segment. Set to 0.0 if no bladeless hub segment is to be generated.

NELH = Number of elements in circumferential direction for the bladeless hub segment. Set to 0 for no bladeless hub segment.

Note: See Figure B.7 for definition of variables.

Card 7 (8E10.3) For IFR = 1 or 3 or alternately IFR = 2 when ICF ≠ 0 (as may be required).

$CP_{(1)}$ = Chord fractions for fillet data definition along propeller blade.
↓
$CP_{(NCST)}$

If IFR = 0, include only Card 8.
= 1, include only Cards 9 and 10.
= 2, include only Card 11.
= 3, include only Cards 12 to 17, inclusive.

Card 8 (E10.3)

FR = Constant fillet radius (length) (same for face and back).

Card 9 (8E10.3)

$FRF_{(1)}$ = Fillet radius (face) at chord positions 1 through NCST (length).
↓
$FRF_{(NCST)}$

Card 10 (8E10.3)

$FRB_{(1)}$ = Fillet radius (back) at chord positions 1 through NCST (length).
↓
$FRB_{(NCST)}$

Card 11 (3E10.3) (see Figure B.3)

AK1 = Constant factor for two radii fillet surface. Defaults to 1.0

AK2 = Constant factor for two radii fillet surface. Defaults to 3.0

AK3 = Constant factor for two radii fillet surface. Defaults to 1/3.

Card 12 (8E10.3)

$AK1F_{(1)}$ = Constant factor (face) for two radii fillet surface at chord positions 1 through NCST.
↓
$AK1F_{(NCST)}$

Card 13 (8E10.3)

$AK2F_{(1)}$ = Constant factor (face) for two radii fillet surface at chord positions 1 through NCST.
$\downarrow$
$AK2F_{(NCST)}$

Card 14 (8E10.3)

$AK3F_{(1)}$ = Constant factor (face) for two radii fillet surface at chord positions 1 through NCST.
$\downarrow$
$AK3F_{(NCST)}$

Card 15 (8E10.3)

$AK1B_{(1)}$ = Constant factor (back) for two radii fillet surface at chord positions 1 through NCST.
$\downarrow$
$AK1B_{(NCST)}$

Card 16 (8E10.3)

$AK2B_{(1)}$ = Constant factor (back) for two radii fillet surface at chord positions 1 through NCST.
$\downarrow$
$AK2B_{(NCST)}$

Card 17 (8E10.3)

$AK3B_{(1)}$ = Constant factor (back) for two radii fillet surface at chord positions 1 through NCST.
$\downarrow$
$AK3B_{(NCST)}$

Card 18 (8E10.3) For ICF $\neq 0$, only.

$FBITL_{(1,1)}$ = Length of tangent vector at chord positions 1 through NCST (face) on the fillet to blade intersection (tangency) line. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. (See Note on continuous function fillet.)
$\downarrow$
$FBITL_{(NCST,1)}$

Card 19 (8E10.3) For ICF $\neq 0$, only.

$FBITL_{(1,2)}$
↓
$FBITL_{(NCST,2)}$ = Length of tangent vector at chord positions 1 through NCST (back) on the fillet to blade intersection (tangency) line. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. (See Note on continuous function fillet.)

Card 20 (8E10.3) For ICF $\neq 0$, only.

$FHITL_{(1,1)}$
↓
$FHITL_{(NCST,1)}$ = Length of tangent vector at chord positions 1 through NCST (face) on the fillet to hub intersection (tangency) line. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. (See Note on continuous function fillet.)

Card 21 (8E10.3) For ICF $\neq 0$, only.

$FHITL_{(1,2)}$
↓
$FHITL_{(NCST,2)}$ = Length of tangent vector at chord positions 1 through NCST (back) on the fillet to hub intersection (tangency) line. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. (See Note on continuous function fillet.)

Card 22 (3E10.3)

DR = Fraction of full blade radius used in iteration procedure for locating fillet. Defaults to 0.01.

DANG = Angle increment (degrees) used in integration procedure for locating fillet. Defaults to 1.0.

TOL = Tolerance to which iteration is carried out. Defaults to 0.001.

Note on Continuous Function Fillet

The continuous function fillet generation capability provides a more flexible fillet surface description without compromising smoothness as in the case of the two radii fillet option.

At a typical chord position, the continuous function fillet is described by a cubic parametric spline of form:

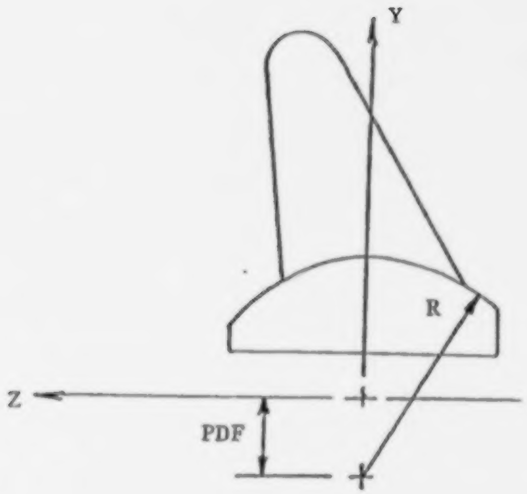
$$\begin{Bmatrix} X \\ Y \\ Z \end{Bmatrix} = \begin{bmatrix} a_o & a_1 & a_2 & a_3 \\ b_o & b_1 & b_2 & b_3 \\ c_o & c_1 & c_2 & c_3 \end{bmatrix} \begin{Bmatrix} 1 \\ \alpha \\ \alpha^2 \\ \alpha^3 \end{Bmatrix}$$

where the parameter α varies between 0 at the tangency point of the blade-to-fillet intersection (FBI) and 1 at the tangency point of the hub-to-fillet intersection (FHI).

Unless the tangent vector length $ds/d\alpha$ is also defined at the FBI and FHI points, the parametric derivatives of the functions defining coordinates X,Y and Z for the spline curve are incompletely defined. While the notion of a tangent vector length is somewhat abstract, it does have a significant influence on the fillet geometry generated (as demonstrated in Figure B.8) with values generally ranging from 0.1 to 10.0 times the distance between the FBI and FHI points.

Due to possible difficulties in estimating tangent vector lengths for input to the program, provisions have been made for their automatic computation from a best curve fit to the two radii fillet. When zeros are supplied for tangent vector lengths at both FBI and FHI points on any chordwise positions of either face or back surfaces, an iterative curve fitting routine is activated. (The calculated tangent lengths are printed for possible use in subsequent runs.) When the chordwise data stations specified in CARD 7 do not correspond to those of the finite element grid (specified in CARDS 13 to 15 of Appendix B.1), it is advised that the curve fitting feature be utilized for either none or all chordwise positions.

Spherical
Palm Only



General

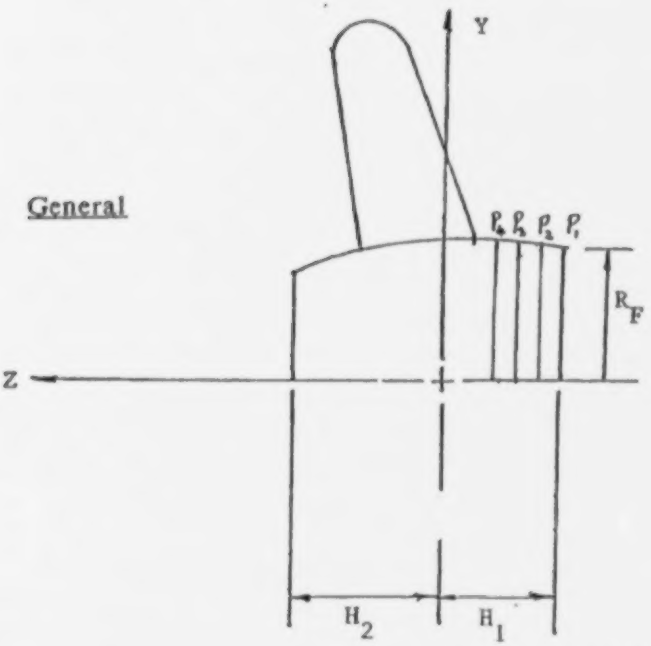

Y
ρ_4 ρ_3 ρ_2 ρ_1
R_F
Z
H_2
H_1

FIGURE B.2: Types of Blade Palm/Hub Models

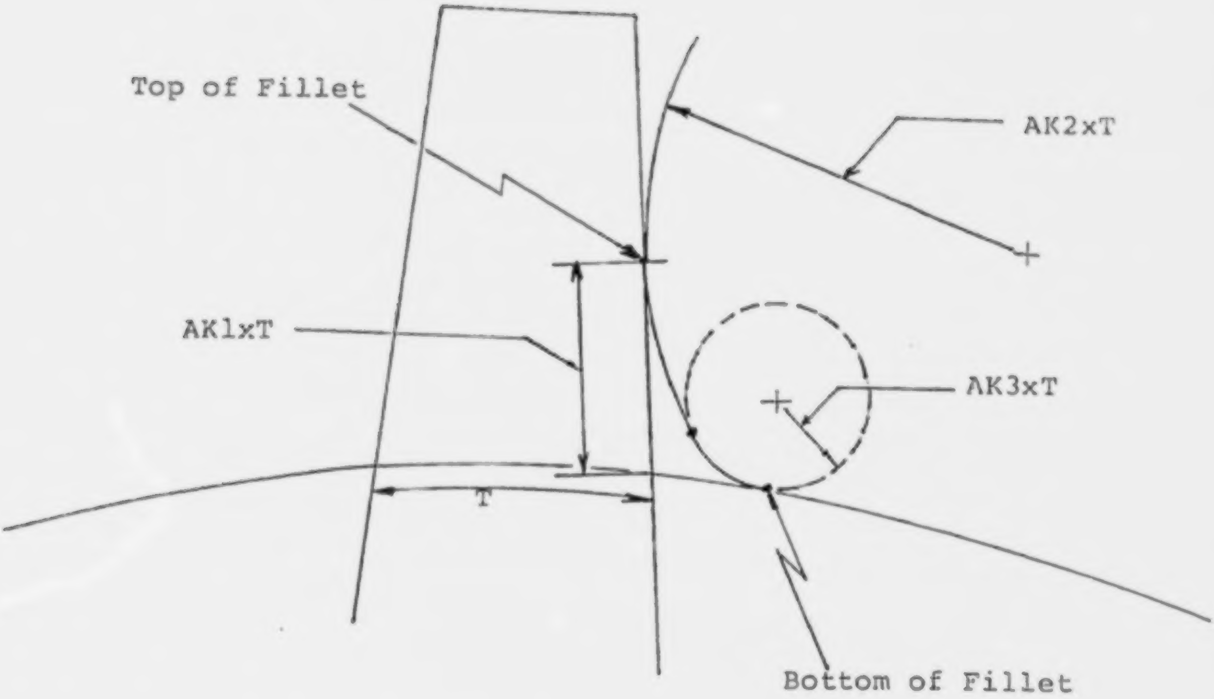
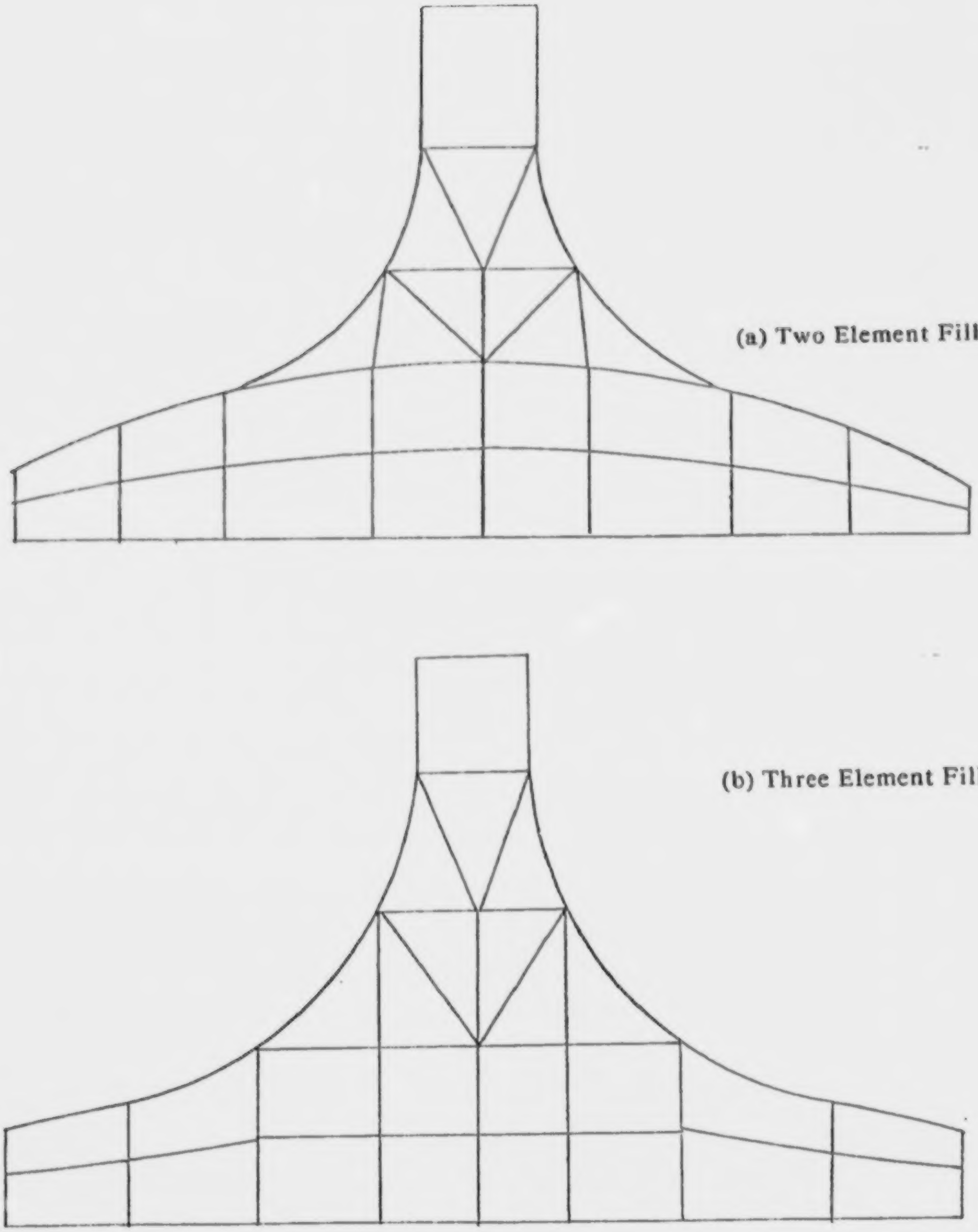

Top of Fillet
AK2xT
AK1xT
AK3xT
T
Bottom of Fillet

FIGURE B.3: Two Radii Fillet at Root



(a) Two Element Fillet

(b) Three Element Fillet

FIGURE B.4: Fillet Representation

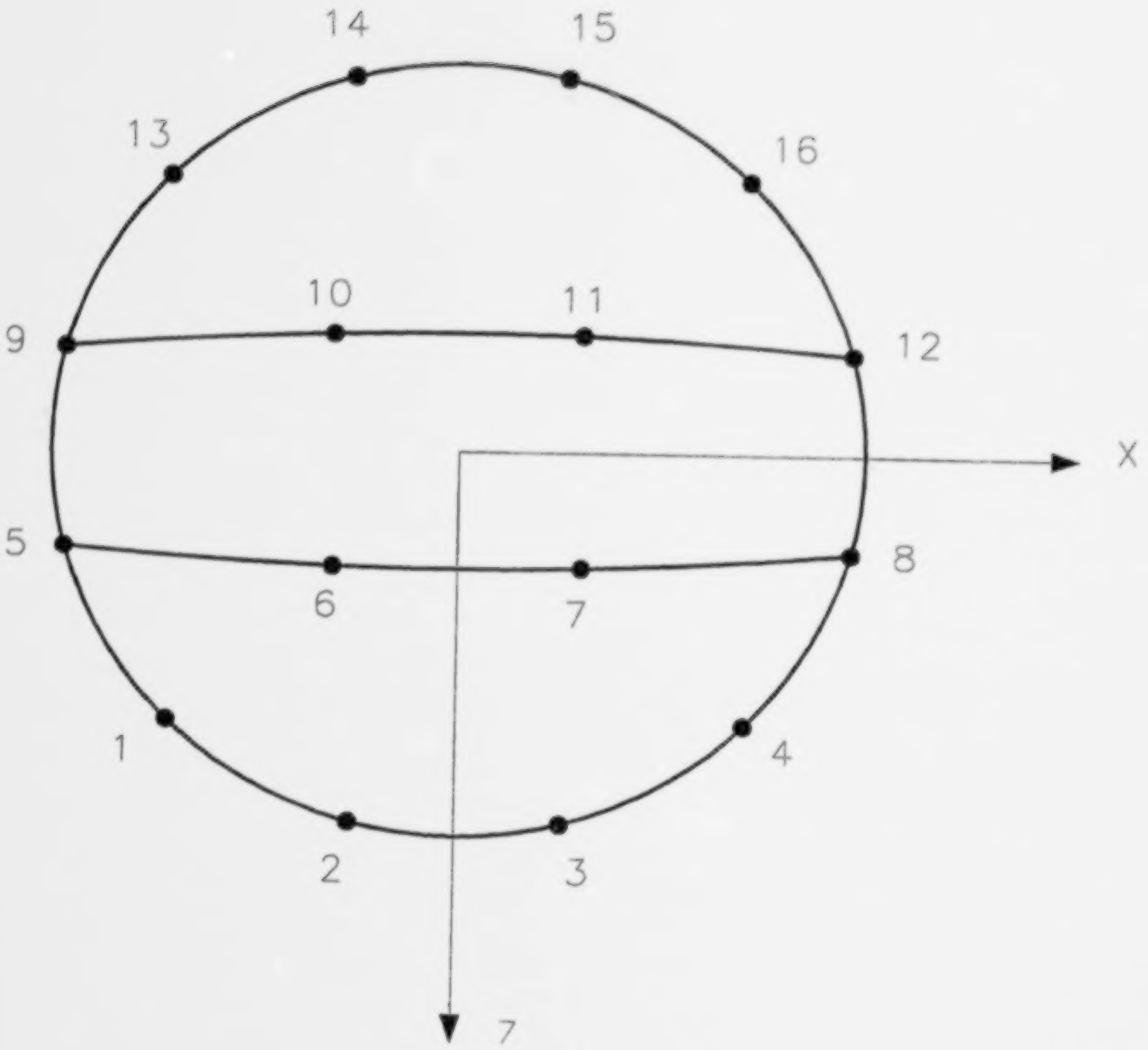

14
15
13
16
10
11
9
12
X
5
8
6
7
1
4
2
3
7

FIGURE B.5: Location of 16 Points Defining Patch for Modelling Palm Surface

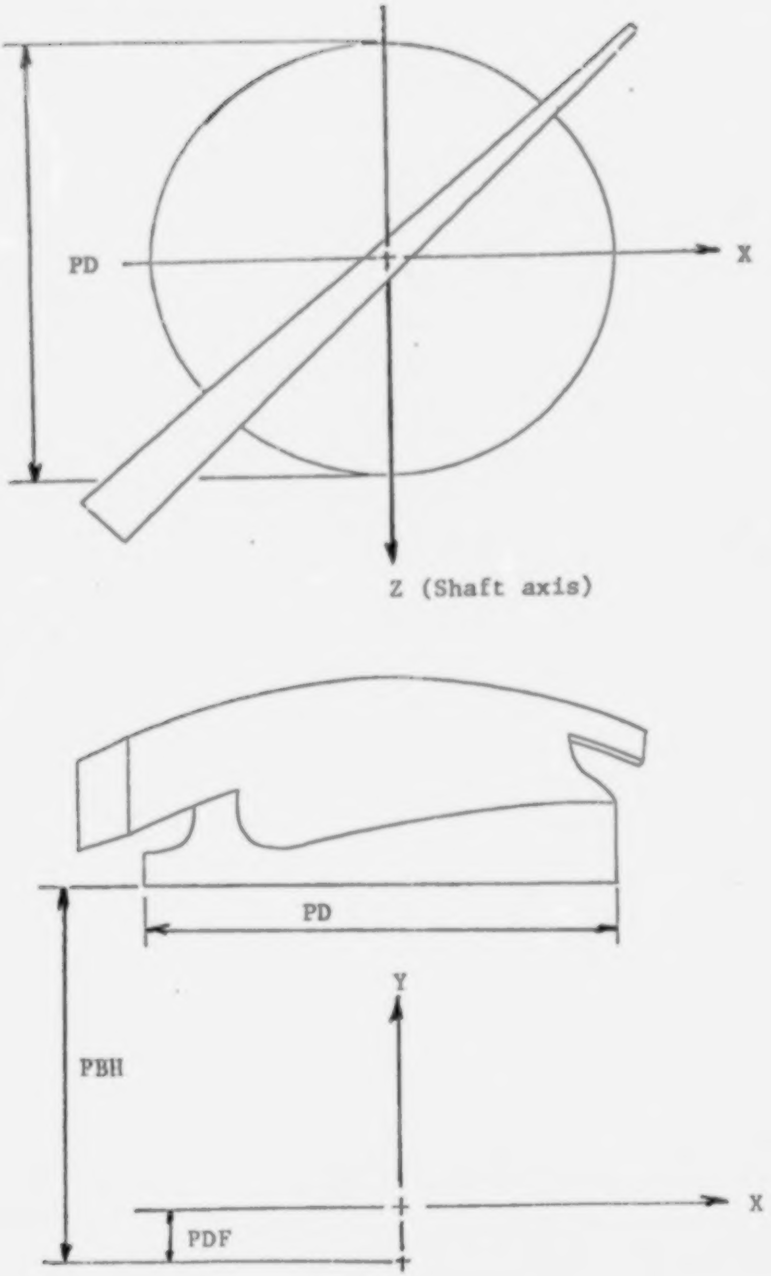
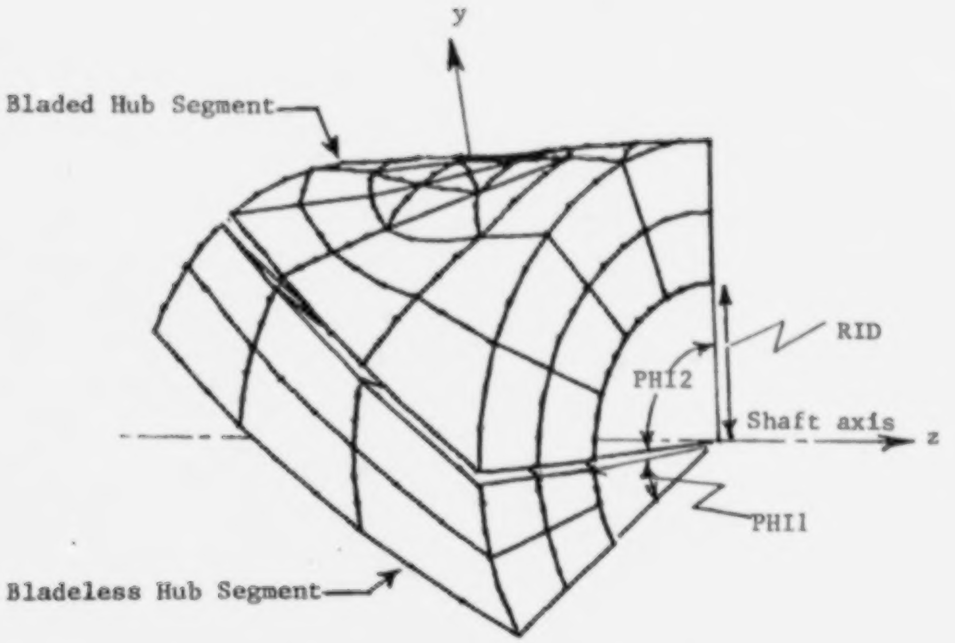

PD
X
Z (Shaft axis)
PD
Y
PBH
X
PDF

FIGURE B.6: Geometric Parameters for Palm Model


y
Bladed Hub Segment
RID
PHI2
Shaft axis
z
PHI1
Bladeless Hub Segment

PHI = PHI1 + PHI2

PHI1 = PHI * PHIF

FIGURE B.7: Geometric Parameters for Hub Model

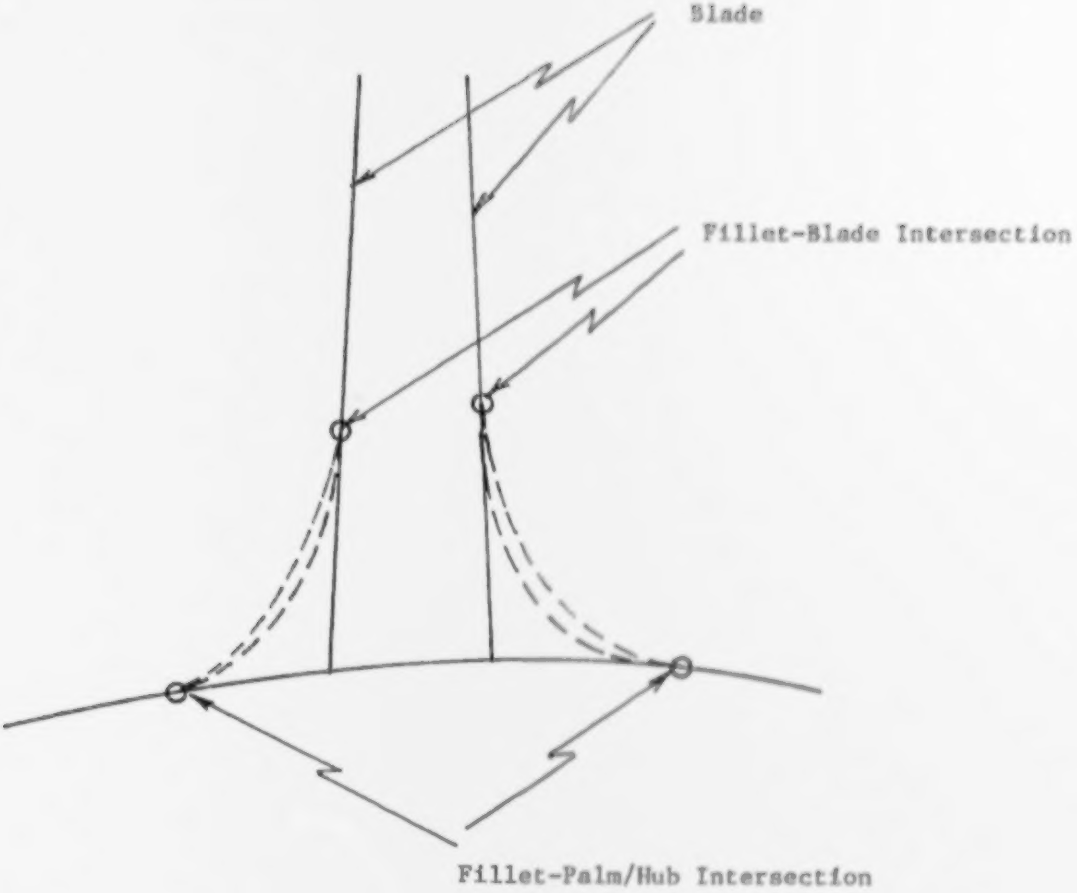

Blade
Fillet-Blade Intersection
Fillet-Palm/Hub Intersection

FIGURE B.8: Plot Showing How Different Vector Lengths Will Vary Fillet Shape

B.3 PVBLD2: Generation of finite element model for propeller blade.

Header Card (A6)

BLADE2 (See Appendix K for further details on use of header cards.)

Card 1 (4I5)

IGRID = 1, grid #1 is generated (see Figure B.9).
2, grid #2 is generated (see Figure B.9).

Note: In order to model blade as a superelement, IGRID must be equal to 2.

IBASE = 0, first radial position taken as base of blade.
= 1, palm/hub model is used to generate simple blade fillet root on base of blade. The fillet region is modelled with one element through the thickness to provide boundary condition attachment at base.
= 2, palm/hub model is used to generate bottom of blade base. The blade model begins at the top of the fillet (substructuring only).

Note: In order to model blade as a superelement, IBASE must be equal to 2.

IELEM = 0, shell elements are used to model blade.
= 1, 20-noded elements are used to model blade.

IPTC = 0, printout (of nodal coordinates, elements, grid) is reduced.

Note: If IELEM=0 and IBASE=2, the program replaces the bottom row of shell elements by transition elements.

Card 2 (3E10.3)

E = Young's Modulus (force/length^2).

PR = Poisson's ratio.

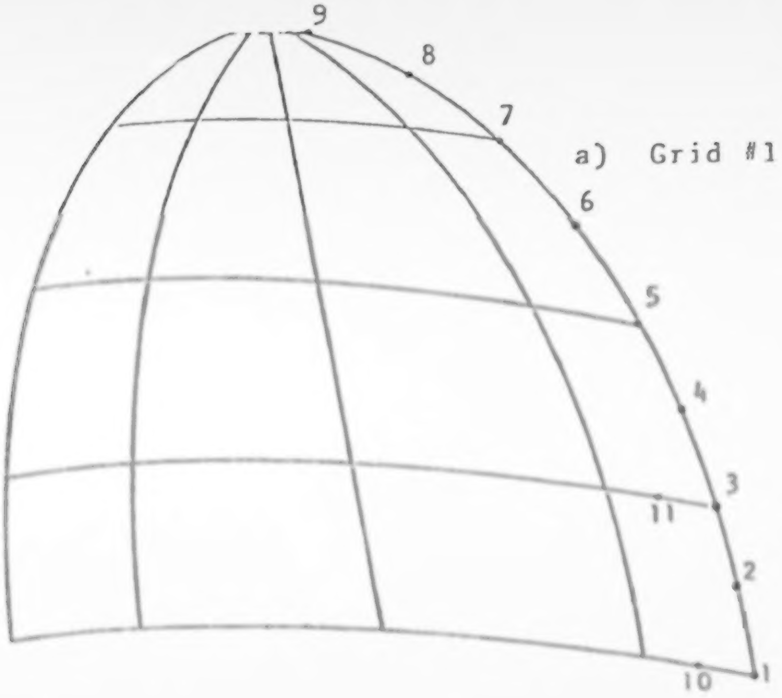
DEN = Material weight per unit volume/g (force-time^2/length^4).

Card 3 (I5) Omit if IELEM ≠ 1

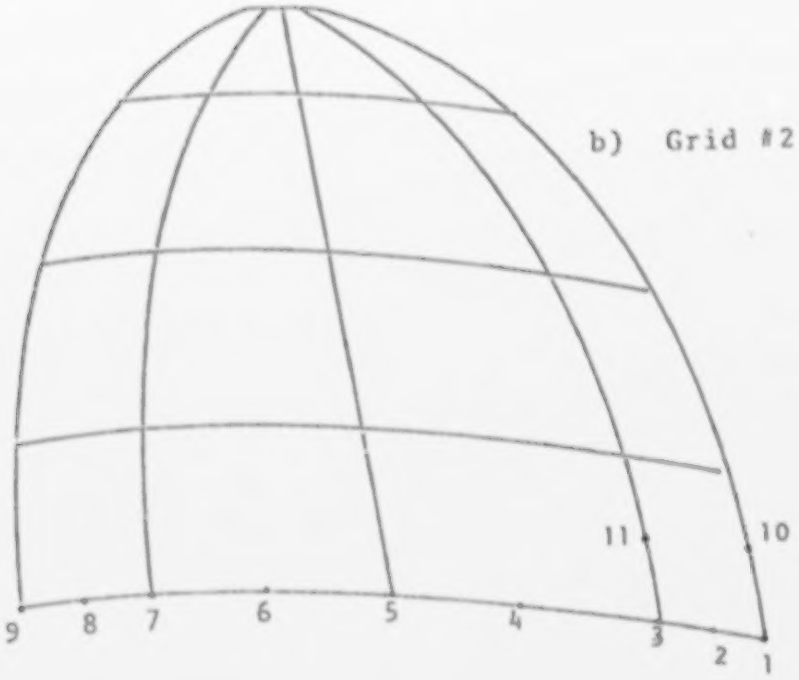
IWDGE = Parameter for collapsing 20-noded solid elements to wedges at the leading and trailing edges.

= 0 no wedges are formed.

= 1 wedges are formed.


9
8
7
6
5
4
3
11
2
10
1

a) Grid #1


11
10
9
8
7
6
5
4
3
2
1

b) Grid #2

FIGURE B.9: Node Numbering Schemes for Blade Model

B.4 PVBLD3: Generation of blade model using a graphics tablet.

This program provides the user with the capability to interactively describe a finite element mesh for a blade via free-hand drawing using a graphics tablet. The necessary requirements are: (1) a 4010-Series Tektronix Computer Display Terminal, (2) a Tektronix 4953 or 4954 Graphics Tablet, and (3) a scaled true space outline of the x-y plot (or transverse projection).

The basic preparatory steps are as follows:

1. Sketch the desired finite element grid on the scaled transverse projection. A typical transverse projection is shown in Figure B.10(a), and a typical finite element grid is shown in Figure B.10(b).
2. Designate and number the displacement nodes, i.e. corner and mid-side nodes [see Figure B.10(b)].
3. Define a scale as shown in Figure B.10(b).
4. Tape the diagram onto the surface of the graphics tablet. No special orientation is necessary.

The writing implement is either a pen or an optional flat cursor with a button, and is connected to the power module. The power module is a box containing control circuitry for the tablet and writing implement, an ON/OFF switch, and power supply. In addition, three red lights on the front of the box indicate when the circuitry is READY to send points, when the DATA is being digitized and transmitted, and when the PEN (or cursor) is depressed.

Subprogram PVBLD3 has not been implemented into the PVAST overlay and must therefore be operated in the standalone mode (MODE=1, See p5.1-1). Once the preparatory steps have been taken, the display terminal signed on, and the power module ON/OFF switch in the ON position, the user should proceed as follows:

```
LOAD PVBLD3,<LIB>TAB,<LIB>PLOT10/LIB
START
```

Terminal Response:

(1) SPECIFY YOUNG'S MODULUS, POISSON'S RATIO AND DENSITY.

Enter, in free format, values for Young's Modulus, Poisson's ratio, and density.

(2) SPECIFY NUMBER OF DISPLACEMENT NODES (NDN) AND ELEMENTS (NELM).

Enter number of displacement nodes and number of elements in free format. This information must be established by the user prior to executing PVBLD3.

(3) SPECIFY SCALE.

Enter scale in free format, see Figure B.10(b). The unit for the scale is inches.

The terminal is now ready to accept a series of data points from the graphics tablet. The READY light on the power module should be on. Each data point is sent to the terminal by first depressing the pen (or pushing the bottom of the flat cursor) and then pressing the RETURN key on the terminal keyboard. Every time the READY light on the power module comes on, the terminal is ready to accept a single data point. A description of the points to be sent to the terminal is as follows:

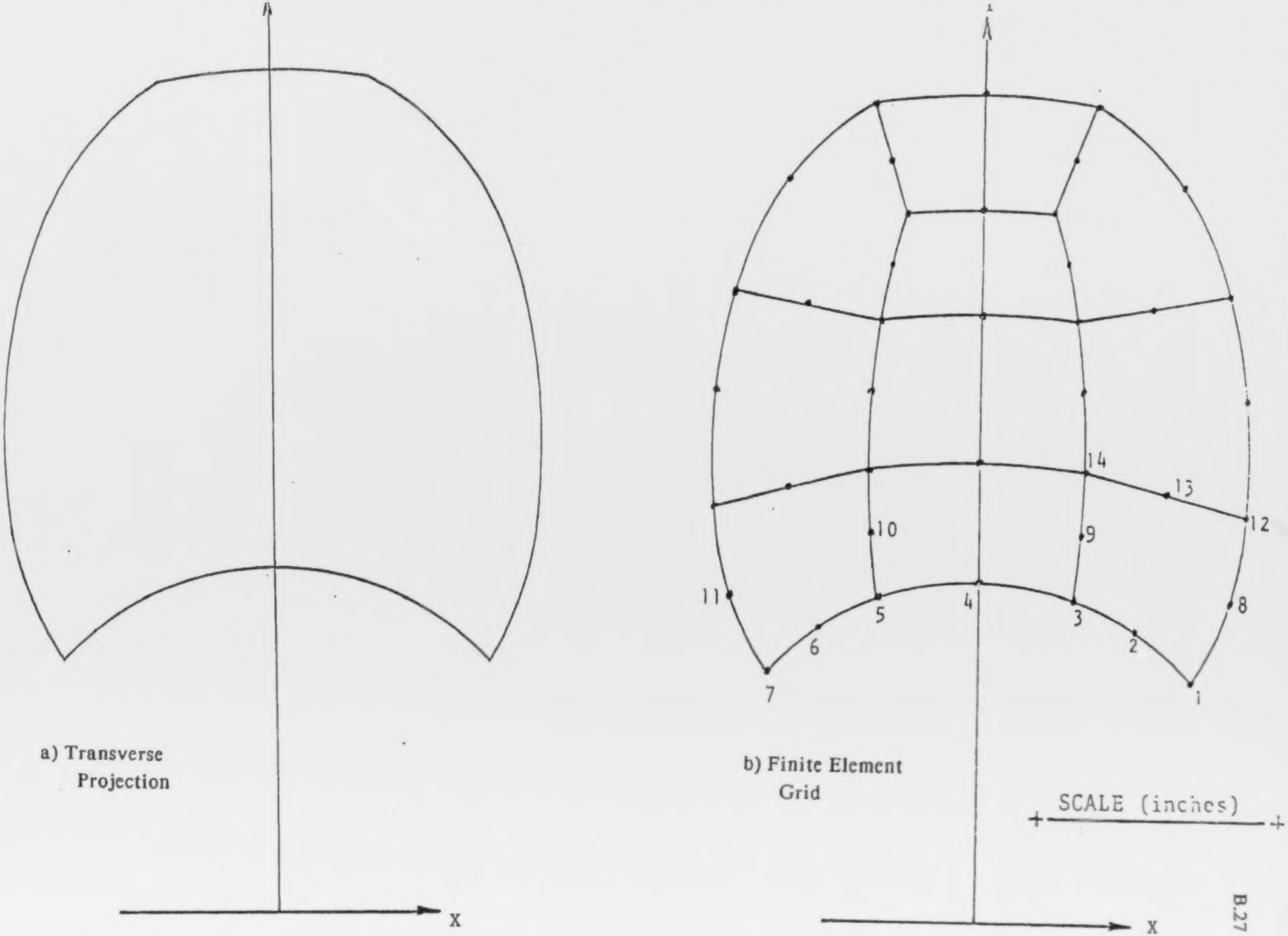
First three - (1) lower left hand corner, (2) lower right hand corner of drawing, and (3) upper right hand corner. These three points are used to define boundary and scale of the terminal screen.

Next two - (1) origin of coordinate system, i.e. x=y=0, and (2) a point at the blade top on the y axis. When the second point has been sent, the y axis should appear on the terminal screen.

Next two - (1) extreme left hand point, and (2) extreme right hand point of the drawing scale. When the second point has been sent, the scale should be drawn on the terminal screen.

Next NDN - The displacement nodes. These nodes are numbered in the order in which they are sent to the terminal. Each point should appear on the terminal screen directly after it has been sent. If for some reason a point was sent incorrectly, it can be sent again by preceding it with a point which is outside the drawing boundary (if a point continues to be sent incorrectly, it is probably due to a null spot on the tablet; and, rather than reposition the diagram on the tablet and restart the data transmission, it is usually best to move this displacement node slightly away from its original position).

Next 8xNELM - The element connectivities. Eight nodes are required for each element. A node is defined by sending a point which is close to the corresponding displacement node. One element must be completed before the next one is started. The nodes for a particular element are sent in the following order (see Figure 3.1): N1,N5,N2,N6,N3,N7,N4, and N8. After a group of eight nodes has been sent, an outline of the element should appear on the terminal screen. If for some reason the nodes of the element were not defined correctly, they can be redefined by preceding the first node with a point which is outside the drawing boundary. The process of defining the element nodes (and, in fact, the entire operation) is complete when the message "PREFX.T10 HAS BEEN CREATED" appears on the terminal screen.


X
1
2
3
4
5
6
7
8
9
10
11
12
13
14
SCALE (inches)
X

a) Transverse Projection

b) Finite Element Grid

FIGURE B.10: Typical Drawings Required for Describing a Finite Element Mesh Using a Graphics Tablet

B.5 PVSUB1, PVSUB2: Generation of substructured finite element model of combined blade, fillet, hub or palm and optionally bolts as rods or bolts and bolthole substructures modelled in detail.

Header Card (A6)

SUBMOD (See Appendix K for further details on use of header cards.)

Card 1 (3I5)

NBLADE = Number of blades. (For palm model, only one blade should be specified.)

IRMN = Root master node code. If IRMN = 0, only master nodes at interfaces between blade and fillet as well as between fillet and palm/hub root are determined. If IRMN = 1, master nodes at interfaces between blade and fillet, at the interface between the fillet and palm/hub root, and along the sides of the hub are determined. This is necessary when modelling the complete hub.

IHSS = Superelement generation code for bladeless hub segment. Set ≠0 if bladeless hub segment(s) is to be included in the hub model.

* Provide Cards 2 and 3 for NBLADE blade(s).

Card 2 (I5)

NUSMNB = Number of user specified master nodes for the blade (in addition to the ones at the blade/fillet interface). If NUSMNB = -1, user specified master nodes for previous blade are used.

Card 3 (16I5) Omit, if NUSMNB = 0 or -1.

$IMN_{(1)}$ = User specified master nodes.
↓
$IMN_{(NUSMNB)}$

* Provide Cards 4 and 5 for NBLADE fillet section(s).

Card 4 (I5)

NUSMNF = Number of user specified master nodes for the fillet section (in addition to the ones at the blade/fillet and fillet/palm/hub interface). If NUSMNR = -1, user specified master nodes for previous fillet section are used.

Card 5 (16I5) Omit, if NUSMNF = 0 or -1.

$IMN_{(1)}$ = User specified master nodes.
↓
$IMN_{(NUSMNF)}$

* Provide Cards 6 and 7 for NBLADE palm or bladed hub section(s).

Card 6 (I5)

NUSMNR = Number of user specified master nodes for the palm/hub section (in addition to the ones at the fillet/palm/hub interface and on the sides of the hub). If NUSMNR=-1, user specified master nodes for previous palm/hub sections are used.

Card 7 (6I5) Omit, if NUSMNR = 0 or -1.

$IMN_{(1)}$
↓ = User specified master nodes.
$IMN_{(NUSMNR)}$

* Provide Cards 8 and 9 for NBLADE bladeless hub section(s).
Omit if a palm model has been generated or if IHSS=0.

Card 8 (I5)

NUSMNI = Number of user specified master nodes for the bladeless hub section. If NUSMNI=-1, user specified master nodes for previous section are used.

Card 9 (16I5) Omit, if NUSMNI = 0 or -1.

$IMN_{(1)}$ = User specified master nodes.
↓
$IMN_{(NUSMNI)}$

* Provide Cards 10-13 once for each of the NBOLT pairs of bolt and boltholes (NBOLT defined in PVBLT data). Omit if a palm with detailed bolt and bolthole substructures is not being generated.

Card 10 (I5)

NUSMN1 = Number of user specified master nodes for the bolthole (in addition to the ones at the palm/bolthole and bolthole/bolt interfaces).

Card 11 (16I5) Omit, if NUSMN1 = 1.

$IMN_{(1)}$ = User specified master nodes.
↓
$IMN_{(NUSMN1)}$

Card 12 (I5)

NUSMN2 = Number of user specified master nodes for the bolt (in addition to the ones at the bolthole/bolt interfaces).

Card 13 (16I5) Omit, if NUSMN2 = 0

$IMN_{(1)}$ = User specified master nodes.
↓
$IMN_{(NUSMN2)}$

B.6 PVKEY: Generation of two dimensional finite element models for hubs with keyways and their loads.

Header Card (A6)

KYHUB (See Appendix K for further details on the use of header cards.)

Card 1 (A72)

TITLE = 72 character title.

Card 2 (5E10.3)

RI = Inner radius (length).

RO = Outer radius (length).

A = One half keyway width (length).

B = Keyway depth (length).

HLEN = Hub length (length) (see Figure B.11).

Card 3 (2E10.3,I5)

E = Young's modulus (force/length^2).

PR = Poisson's ratio.

ICODE = 0, plane stress is assumed.
≠ 0, plane strain is assumed.

Card 4 (I5,E10.3)

NBLAD = Numbers of blades on hub.

PHI = Angle in degrees from vertical to spindle axis of first blade (positive clockwise). Angle of first blade must be greater than 90°.

Card 5 (3E10.3)

FTR = Tangential force at each blade root (positive clockwise).

MZR = Overturning moment at each blade root (positive about z - axis).

FRR = Radial force at each blade root (positive along outward radial direction).

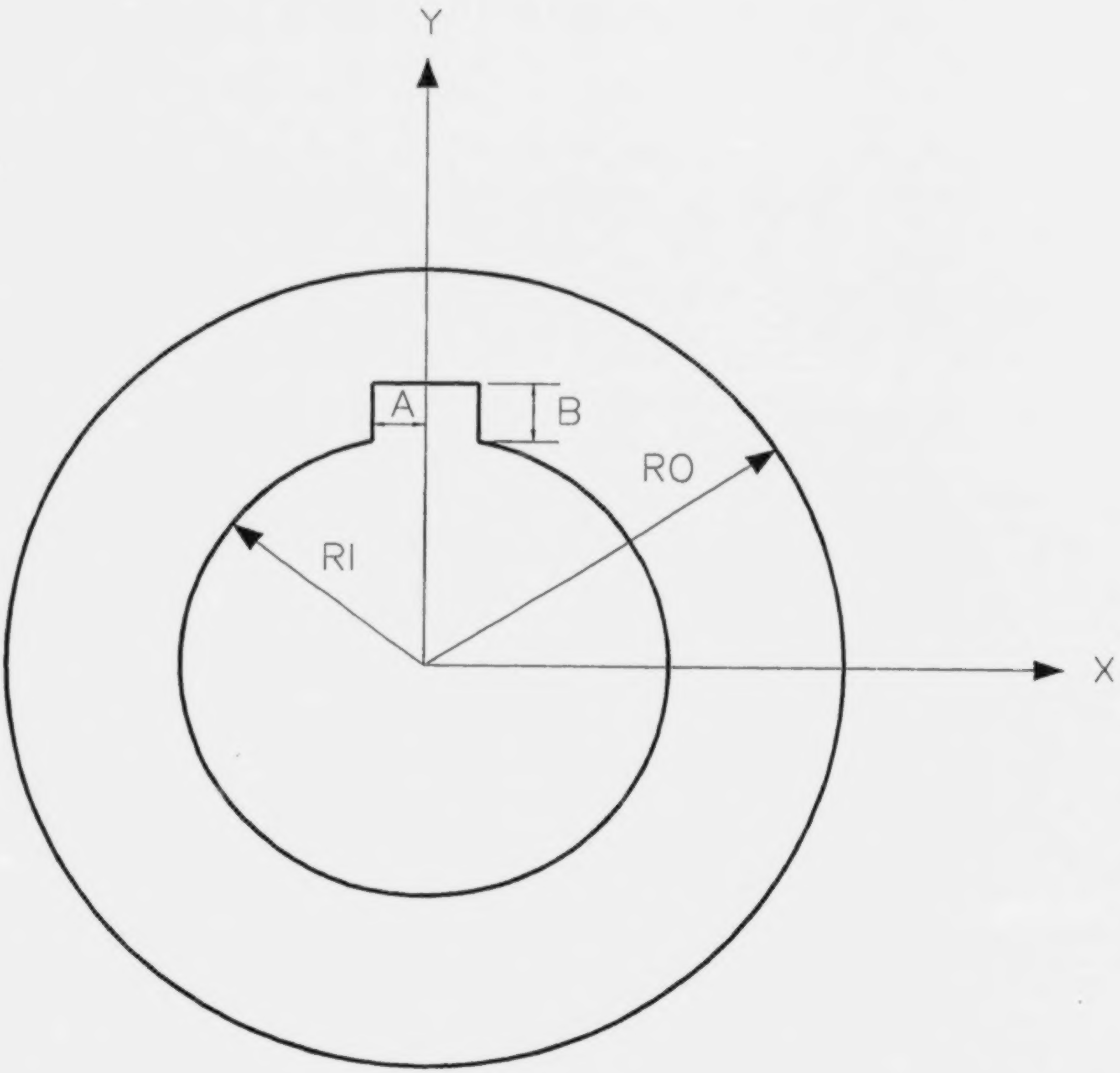


FIGURE B.11: Geometry of the HUB with Keyway

B.7A **PVBCON:** Interactive definition and/or modification of boundary conditions for finite element model.

Terminal Response:

(1) SUBPROGRAM PVBCON PROGRAM TO INTERACTIVELY DEFINE AND/OR MODIFY BOUNDARY CONDITIONS FOR FINITE ELEMENT MODEL.

(2) A FILE OF BOUNDARY CONDITIONS EXISTS.
ENTER C TO CONTINUE AND REGENERATE THE DATA.
N TO NOT REGENERATE IT:

The above prompt and entry occur only if a boundary condition file named PREFX.TBC exists on the user connected directory. Entering C permits the file to be modified or a new boundary condition file to be generated independently and substituted in its place. Control is transferred to step (13). Entering N on the other hand leads to termination of the subprogram with the original boundary condition file intact.

(3) SHOULD OLD BOUNDARY CONDITIONS SERVE AS STARTING POINT FOR REGENERATIONS? ENTER N OR Y:

If required model boundary conditions resemble sufficiently the old boundary conditions on file PREFX.TBC, enter Y to begin modifying them. Control is then transferred to step (13).

(4) SHOULD DEFAULT BOUNDARY CONDITIONS FOR THIS STRUCTURE ON FILE PREFX.T10 SERVE AS STARTING POINT FOR REGENERATION? ENTER N OR Y:

For certain types of propeller models a set of default boundary conditions are automatically generated and stored on files describing the finite element model. Only for those models is a prompt similar to the above produced. Frequently the default boundary conditions will satisfy the user requirements with little or no modification and in that case Y is the appropriate response.

(5) WILL BOUNDARY CONDITIONS BE SPECIFIED BY RESPONSE TO INTERACTIVE TERMINAL PROMPTS? ENTER N OR Y:

Should neither an old boundary condition file nor default boundary condition data set exist or alternately the user considers neither a suitable basis for model boundary condition definition, boundary conditions should then be defined in response to interactive terminal prompts. This offers not only the convenience of free format data input but also other features to reduce input requirements when several nodes have the same constraint specification.

(6) ENTER DEFAULT VALUE OF BOUNDARY SPRINGS:

Specify default value of boundary spring (see page C6-4 of the VAST user manual) in free format.

(7) ENTER NUMBER OF FIRST CONSTRAINED NODE:

Specify number of first node with constraints applied.

(8) SPECIFY BOUNDARY CONDITION CODES FOR NODAL FREEDOMS IN FREE FORMAT (SEE PAGE C6-4 OF THE V-A-S-T MANUAL) OR ALTERNATELY S IF SAME AS FOR THE LAST NODE:

Specify boundary condition codes for nodal freedoms on one line in free format.

(9) SPECIFY BOUNDARY SPRINGS IN FREE FORMAT (SEE PAGE C6-4 OF THE V-A-S-T MANUAL) OR ALTERNATELY S IF SAME AS FOR THE PREVIOUS NODE:

Specify boundary springs on one line in free format.

(10) ENTER NEXT CONSTRAINED NODE (ZERO TO TERMINATE INPUTS):

(11) SPECIFY BOUNDARY CONDITION CODES (OR S):

(12) SPECIFY BOUNDARY SPRINGS (OR S):

Repeat steps (10) to (12) for remaining constrained nodes within the structure. Terminate input by entering zero in response to the node number prompt.

(13) SHOULD CURRENT BOUNDARY CONDITIONS BE LISTED? ENTER N OR Y:

Respond appropriately depending on whether a terminal listing of the current status of the boundary condition file is required.

(14) SHOULD VALUE OF DEFAULT SPRING BE ALTERED? ENTER N OR Y:

Answering in the affirmative will be followed by a prompt for the new value of the default spring. Program defined default spring specifications generally do not have to be altered unless the structure is extremely stiff and typical stiffness coefficients are significantly larger than 10^{10}.

(15) NODAL CONSTRAINT. CAN NOW BE MODIFIED USING THE FOLLOWING TYPICAL INSTRUCTIONS:

P**: PRINT CONSTRAINT CONDITION FOR NODE **.
M**: MODIFY CONSTRAINT CONDITION FOR NODE **. CURRENT CONSTRAINT CONDITIONS FOR THE NODE ARE FIRST PRINTED AND THEN THE USER CAN ALTER THEM BY TYPING REQUIRED MODIFICATIONS IMMEDIATELY BELOW.
I**: INSERT CONSTRAINT CONDITIONS FOR NODE **.
D**: DELETE CONSTRAINT CONDITIONS FOR NODE **.
C**,$$: COPY CONSTRAINTS OF NODE ** INTO NODE $$.
E: END THE CONSTRAINT MODIFICATION SESSION.
H: HELP. PRINTOUT AVAILABLE OPTIONS.

Several command options are available to facilitate boundary condition modification. The list is printed once at the beginning of the constraint modification session unless it is requested through the HELP command. Constraint modifications are printed at the terminal following the input of each command and should serve as prompt for the next command. To end the constraint modification session enter E.

(16) BOUNDARY CONDITION FILE CALLED PREFX.TBC HAS BEEN CREATED.

The name of the boundary condition file is identified.

B.7B PVGAP: Consolidation into a single file of all gap element information for the finite element model defined automatically by programs or optionally defined interactively by user in earlier program steps but generally stored on several independent data files with provisions for final interactive modification.

Terminal Response:

(1)

```
SUBPROGRAM PVGAP    PROGRAM TO INTERACTIVELY DEFINE AND/OR
                    MODIFY GAP ELEMENTS FOR FINITE ELEMENT
                    MODEL.
```

(2)

```
A FILE OF GAP ELEMENTS EXISTS.
ENTER  C TO CONTINUE AND REGENERATE THE DATA.
       N TO NOT REGENERATE IT:
```

The above prompt and entry occur only if a gap element file named PREFX.GAP exists on the user connected directory. Entering C permits the file to be modified or a new gap element file to be generated independently and substituted in its place. Control is transferred to step (13). Entering N on the other hand leads to termination of the subprogram with the original gap element file intact.

(3)

```
SHOULD OLD GAP ELEMENTS SERVE AS STARTING POINT FOR
REGENERATIONS? ENTER N OR Y:
```

If required model gap elements resemble sufficiently the old gap elements on file PREFX.GAP, enter Y to begin modifying them. Control is then transferred to step (13).

(4)

```
SHOULD GAP ELEMENTS FOR THIS STRUCTURE ON FILE PREFX.PLM SERVE
AS STARTING POINT FOR REGENERATION? ENTER N OR Y:
```

For certain types of propeller models a set of default gap elements are automatically generated and stored on files describing the finite element model. Only for those models is a prompt similar to the above produced. Frequently the default gap elements will satisfy the user requirements with little or no modification and in that case Y is the appropriate response.

(5) WILL GAP ELEMENTS BE SPECIFIED BY RESPONSE TO INTERACTIVE TERMINAL PROMPTS? ENTER N OR Y:

Should neither an old gap elements file nor default gap elements dataset exist or alternately the user considers neither a suitable basis for model gap element definition, gap elements should then be defined in response to interactive terminal prompts. This offers not only the convenience of free format data input but also other features to reduce input requirements when several nodes have the same constraint specification.

(6) SHOULD FORCE AND MOMENT RESIDUALS BE COMPUTED. ENTER N OR Y:

Force and moment residuals permit numerical errors generated during equilibrium equation solution to be estimated.

(7) DO YOU WISH TO REMOVE ALL GAP ELEMENTS. ENTER N OR Y:

If examination of the gap elements indicates most are inappropriate, the entire set can be purged by responding in the affirmative.

(8) MAXIMUM NUMBER OF ITERATIONS = XXX
ENTER NEW VALUE. (OR 0 FOR NO CHANGE)

An iterative solution method is currently used in VAST to solve contact problems involving gap elements. The system loads are applied and displacements obtained using conventional solution algorithms for static problems. Components of the displacement vector are then examined for contact between gap element node pairs in the generalized sense that

$$\delta_{1i} - \delta_{2i} - G_i > 0$$

By partitioning the assembled stiffness matrix, a set of equations is obtained which define gap element forces to produce zero gap width. If any gap forces are negative (tensile) the process of estimating gap element forces is repeated until only compressive gap forces are obtained from the equation system. A new load vector can then be generated and the equilibrium equations solved for a new set of nodal displacements. The whole process is repeated until $\delta_{1i} - \delta_{2i} - G_i < TOL$, $i=1, \ldots N$ where N is the total number of gap elements or upper limit on number of iterations is exceeded.

(9)

```
TOLERANCE TO WHICH ITERATIONS WILL BE CARRIED OUT = X.XXXE+XX
ENTER NEW VALUE. (OR 0 FOR NO CHANGE)
```

Because of numerical round off errors, convergence to the desired gap conditions generally cannot be achieved exactly. Consequently convergence to gap conditions to within the user defined tolerance TOL (see previous note) is considered adequate.

(10)

```
SHOULD CURRENT GAP ELEMENTS BE LISTED. ENTER N OR Y:
```

Respond appropriately depending on whether a terminal listing of the current status of the gap element file is required.

(11)

```
BEGIN MODIFICATIONS:

WHEN PROMPTED ENTER THE NUMBER OF THE GAP ELEMENT TO BE
MODIFIED.
THE DATA FOR THAT ELEMENT WILL BE OUTPUT FOLLOWED BY A>.
ENTER THE NEW DATA FOR THE ELEMENT ON ONE LINE
(TO DELETE A GAP ELEMENT, ENTER D.
IF NO CHANGES ARE TO BE MADE, ENTER C.
TO END MODIFICATIONS, ENTER 0 FOR GAP ELEMENT NUMBER.)
```

Note that when data for the current gap element is to be modified, the entire line must be retyped not just the changes. Since new data is read in free format, columnar correspondence with the old data is unnecessary. When a gap element is deleted, numbers associated with all subsequent gap elements are decremented by one. A new gap element can be added to the list by entering a gap element number one larger than the number of gap elements currently in the model.

(12)

```
ENTER GAP ELEMENT #
```

Specify the number of a gap element requiring modification or creation. If the gap element modification process is complete, enter 0 for transfer of control to (14).

(13)

```
N1   N2  IDC     GAP
 X    X   X   X.XXXE+XX
```

New gap element data supplied is to be entered in free format and columnar correspondences with the line of old data is unnecessary. Enter D to delete the gap element and C to continue to the next element with no modifications. Control returns to (12).

(14) GAP ELEMENT FILE CALLED PREFX.GAP HAS BEEN CREATED.

The name of the gap element file is identified.

B.8 PVBLTA/B/C: Attachment bolt and gap element modelling for palm models.

Two modelling options are available for the investigation of bolt attachment loads and stresses. One is the relatively simple approach of incorporating bolts as rod elements (IOPTB=1) and gap elements optionally at the rod to palm connection as well as at user selected nodes on the blade-carrier interface. In order to use this method, the nodes and elements of the palm finite element model must first be manipulated so that x,z coordinates of each bolt corresponds with a model node. Prestresses can be incorporated to account for the bolt tightening loads. Gap elements as well as constraint boundary conditions are implemented by either cursor selection of nodes or alternately keyboard entry from plots of the palm base with nodes numbered. Subprogram PVBLTA provides this simple palm and bolt modelling capability.

The other option is much more sophisticated and involves detailed substructuring of bolts and bolthole regions (IOPTB=2). Again the mesh in the palm is interactively manipulated. The aim, however, is to achieve suitably shaped elements so that at each bolt location an element or group of elements could be replaced with a bolthole substructure and bolt substructure with mating nodes under the head. This obviously provides a more accurate representation of the attachment of the blade and palm to the crank ring mechanism. Gap element generation on the bolthead to palm interface is controlled by data on the input file but on the blade carrier interface it is interactively controlled by the program user. Boundary conditions are similarly implemented either by cursor selection of nodes or keyboard entry from examination of plots of the palm base with nodes numbered. Subprogram PVBLTB has been developed for this purpose.

Input data requirements for both bolt modelling options are similar and they are outlined below.

Bolt Input Data on CDR File

Header Card (A6)

BOLTS (See Appendix K for further details on use of header cards.)

Card 1 (I5)

NBOLT = Number of bolts through the finite element model of the palm.

Card 2 (3E10.3) (Omit if NBOLT = 0)

BE = Young's Modulus of bolt material (force/length^2).

BPR = Poisson's ratio of bolt material (not required for bolt modelling option IOPTB=1, may be defined as 0.0)

BDEN = Material weight per unit volume/g (force-time^2/length^4).

Card 3 (7E10.3,I5) (Omit if NBOLT = 0)

XCOR, ZCOR = Coordinates of bolt center(length, length).

SDIA = Bolt shaft diameter (length).

SLEN = Bolt shaft length (length).

HDIA = Bolt head diameter (length). (Not required for bolt modelling option IOPTB=1, may be defined as 0.0.)

HLEN = Bolt head thickness (length). (Not required for bolt modelling option IOPTB=1, may be defined as 0.0.)

PSTRES = Bolt prestress (force/length^2). Positive for tension and negative for compression. (Not required for bolt modelling option IOPTB=2.)

IGAP ≠ 0, gap element(s) implemented under bolt head.

REPEAT CARD 3 NBOLT TIMES.

B.8A PVBLTA: Plotting the propeller palm base for the interactive relocation of nodes to achieve suitable bolt attachment points and to identify nodes for gap element attachments or boundary condition specifications.

Terminal Response:

(1) SUBPROGRAM PVBLTA PROGRAM TO PLOT THE PROPELLER PALM BASE FOR THE INTERACTIVE RELOCATION OF NODES TO ACHIEVE SUITABLE BOLT ATTACHMENT POINTS AND TO IDENTIFY NODES FOR GAP ELEMENTS OR BOUNDARY CONDITIONS.

(2) (Plot of propeller palm base viewed from below appears on screen with bolt locations marked.)

At this point the program enters an "invisible command mode" during which both visual prompts and screen echo prints of user issued commands are avoided and audio prompts substituted in their place so as to maintain maximum screen tidiness. The invisible command mode has two levels distinguishable by the number of bells sounded in the audio prompts.

Generally a single bell suggests the primary level and input must be an alphanumeric keyboard character followed by "Return". The following list contains valid keyboard command characters (in either upper or lower case) at this level and their associated operations:

M = Marking of nodes
L = Labelling of nodes
W = Windowing of plot
R = Relocating nodes
D = Redrawing propeller palm base
B = Drawing of border around the plot
S = End terminal operation of primary level invisible command mode

All other characters are ignored and program control remains at the primary level invisible command mode.

A two bell audio prompt on the other hand suggests a secondary invisible command mode and is associated exclusively with graphics cursor operations required for either the windowing or node relocation operations. It requires a single alphanumeric keyboard character input which is, with one exception, arbitrary. The "Return" key is not to be used after each alphanumeric input.

Below, the operations available in the invisible command mode are described in detail:

M (Return) - This causes the exact location of each node to be marked with a small circle. This is desirable for node relocation and should be requested before attempting the operation.

L (Return) - This causes each node to be numbered. It is useful when choosing gap elements and although valid, should not be requested when relocating nodes.

W (Return) - The windowing feature is useful when choosing gap elements or relocating nodes if the grid is fine. It permits magnification of a small portion of the original plot, where either node locations or node numbering are unclear because of the proximity of nodes.

When this command is issued the terminal responds with two bells and the cursor appears. The user should first position the cursor in the lower left hand corner of the desired window and press any alphanumeric keyboard character except "S" and then repeat this procedure for the top right hand corner. At this point a box with sides parallel to the screen axes is drawn to enclose the area selected for magnification. The user may change the window before proceeding by entering "W (Return)" and repeating the above steps.

Once the windowing has been completed to satisfaction, the user should press the return button to have the screen cleared and the windowed portion redrawn to fill the whole screen.

R (Return) - Node relocation is also achieved with the cursor, the appearance of which is signalled by two bells. The user positions the cursor on the node to be moved and presses any alphanumeric keyboard character except "S".

Unless the cursor is positioned within 4 raster units of an existing node, a warning message will be displayed requesting more precise cursor positioning. When positioning within the specified tolerance is achieved, a small circle is drawn at the node to signify acceptance. Two bells will then sound with the reappearance of the cursor which the user should reposition at the desired new location for the node. Then as before any alphanumeric character except "S" should be input in response to which a small square will be drawn at the new nodal location adopted. This procedure can be repeated any number of times for the same or different nodes.

The cycle is broken by typing an "S" which returns control to the primary invisible command status.

Two exceptions exist to the basic node selection procedure described above. If a node is positioned within 10 raster units of any bolt, the bolt coordinates are automatically assumed to be the new nodal coordinates. Also, the relocation of boundary nodes is currently not permitted and any attempt to do so will result in a visual warning message.

D (Return) - This command results in the updated version of the propeller palm base being plotted. When plotting is complete, a single bell sounds signifying control is at the primary invisible command status.

B (Return) - This command causes a title to be printed and a border to be drawn.

S (Return) - This command terminates modifications of the propeller palm with the terminal operated in the invisible command mode. Data tapes for subsequent analysis are created. Simultaneously data integrity checks are performed to identify any model deficiencies which might exist. The following visual terminal responses can now be expected.

(3) (Summary of model deficiencies)

Bolts not located on grid points are identified:

BOLT #XX IS NOT ON A GRID NODE.

(4) FILE PREFX.T28 CONTAINING BOLT DATA CREATED.

(5) TYPE 1 TO CONTINUE MODEL MODIFICATIONS;
0 TO END MODEL MODIFICATIONS.

Enter 1 or 0 in free format according to whether model deficiencies in the form of bolts not located at grid points or improper numbers of gap elements remain to be corrected. Entering 1 returns control to the invisible command mode where further node relocation, gap element selection, etc. can be performed. Entering 0, on the other hand, results in program execution proceeding to step (5).

(6) ENTER 1 TO RETAIN ORIGINAL MODEL;
0 TO PROCESS MODIFICATIONS.

If the user wishes to retain the original grid coordinates rather than replace them with the modified model, enter a non-zero number in free format. This will be desirable for instance if through grid modification two nodes are inadvertently given the same position since their subsequent separation would probably prove impossible without time-consuming manual edits of program data. Restarting mesh modifications with the original grid will probably prove easier.

(7) ENTER 1 TO SELECT NODES FOR GAP ELEMENTS AND/OR BOUNDARY CONDITIONS, OTHERWISE ENTER 0.

This option permits nodes on the base of the palm to have gap elements and/or boundary conditions assigned to them. Entering 0 causes execution of this subprogram option to be ended.

(8) (The base of propeller palm viewed from below is displayed on the screen and all nodes marked.)

(9) ENTER 1 TO WINDOW A PORTION OF THIS PLOT, OTHERWISE ENTER 0.

The windowing feature is useful when choosing gap elements or assigning boundary conditions if the grid is fine. It permits magnification of a small portion of the original plot, where either node locations or node numbering are unclear because of the proximity of nodes.

When this feature is activated, the terminal responds with two bells and the cursor appears. The user should first position the cursor in the lower left hand corner of the desired window and press any alphanumeric keyboard character except "S" and then repeat this procedure for the top right hand corner. At this point a box with sides parallel to the screen axes is drawn to enclose the area selected for magnification.

The user should then press the return button to have the screen cleared and the windowed portion redrawn to fill the whole screen.

(10) ENTER 1 FOR KEYBOARD INPUT;
2 FOR INPUT VIA THE GRAPHICS CURSOR.

If 1 is entered, the nodes will be labelled to assist the user in node selection; if 2 is chosen, the graphics cursor appears. In addition all gap elements or boundary conditions defined in earlier runs are shown on the drawing. An "x" is used to designate a gap element whereas a "+" is used for a boundary condition.

The user may then commence selecting nodes using the method chosen above. For the keyboard input method, the prompt ">" will be repeatedly issued and new gap elements or boundary conditions can be entered. To delete a gap element or boundary condition enter the negative value of the node. To end the selection cycle enter 0.

For the cursor selection method, move the cursor to desired node and press any key except "D" or "S". Entering "D" will delete the gap element (upon which a square will be drawn at the node) or boundary condition and "S" will terminate the selection cycle.

(11) ENTER 1 TO CONTINUE SELECTING NODES;
0 TO STOP.

Entering 1 will allow the node selection procedure to be repeated through transfer of control to (6). This is desirable when node selection is accomplished with windowing and only portions of the propeller base can be examined at any one time.

B.8B PVBLTB: Plotting the propeller palm base for the interactive relocation of nodes to achieve suitably shaped elements for replacement with bolthole substructures and connecting bolt substructures. Provisions are also made to select nodes on the propeller palm base to which gap elements will be attached or boundary conditions applied.

Terminal Response:

(1) SUBPROGRAM PVBLTB PROGRAM TO PLOT THE PROPELLER PALM BASE FOR THE INTERACTIVE RELOCATION OF NODES TO ACHIEVE SUITABLY SHAPED ELEMENTS FOR REPLACEMENT WITH BOLTHOLE SUBSTRUCTURES AND CONNECTING BOLT SUBSTRUCTURES.

(2) (Plot of top propeller palm surface viewed from above appears on screen with bolthole counterbores displayed.)

At this point the program enters an "invisible command mode" during which both visual prompts and screen echo prints of user issued commands are avoided and audio prompts substituted in their place so as to maintain maximum screen tidiness. The invisible command mode has two levels distinguishable by the number of bells sounded in the audio prompts.

Generally a single bell suggests the primary level and input must be an alphanumeric keyboard character followed by "Return". The following list contains valid keyboard command characters (in either upper or lower case) at this level and their associated operations:

M = Marking of nodes
L = Labelling of nodes
W = Windowing of plot
R = Relocating nodes
D = Redrawing propeller palm base
B = Drawing of border around the plot
E = Selecting element groups for replacement by bolt and bolthole superelements
S = End terminal operation of primary level invisible command mode

All other characters are ignored and program control remains at the primary level invisible command mode.

A two bell audio prompt on the other hand suggests a secondary invisible command mode and is associated exclusively with graphics cursor operations required for either the windowing or node relocation operations. It requires a single alphanumeric keyboard character input which is, with one exception, arbitrary. The "Return" key is not to be used after each alphanumeric input.

Below the operations available in the invisible command mode are described in detail:

M (Return) - This causes the exact location of each node to be marked with a small circle. This is desirable for node relocation and should be requested before attempting the operation.

L (Return) - This causes each node to be numbered. It is useful when choosing gap elements and although valid, should not be requested when relocating nodes.

W (Return) - The windowing feature is useful when choosing gap elements or relocating nodes if the grid is fine. It permits magnification of a small portion of the original plot, where either node locations or node numbering are unclear because of the proximity of nodes.

When this command is issued the terminal responds with two bells and the cursor appears. The user should first position the cursor in the lower left hand corner of the desired window and press any alphanumeric keyboard character except "S" and then repeat this procedure for the top right hand corner. At this point a box with sides parallel to the screen axes is drawn to enclose the area selected for magnification. The user may change the window before proceeding by entering "W (Return)" and repeating the above steps.

Once the windowing has been completed to satisfaction, the user should press the return button to have the screen cleared and the windowed portion redrawn to fill the whole screen.

R (Return) - Node relocation is also achieved with the cursor, the appearance of which is signalled by two bells. The user positions the cursor on the node to be moved and presses any alphanumeric keyboard character except "S".

Unless the cursor is positioned within 4 raster units of an existing node, a warning message will be displayed requesting more precise cursor positioning. When positioning within the specified tolerance is achieved, a small circle is drawn at the node to signify acceptance. Two bells will then sound with the reappearance of the cursor which the user should reposition at the desired new location for the node. Then as before any alphanumeric character except "S" should be input in response to which a small square will be drawn at the new nodal location adopted. This procedure can be repeated any number of times for the same or different nodes.

The cycle is broken by typing an "S" which returns control to the primary invisible command status.

Two exceptions exist to the basic node selection procedure described above. If a node is positioned within 10 raster units of any bolt, the bolt coordinates are automatically assumed to be the new nodal coordinates. Also, the relocation of boundary nodes is currently not permitted and any attempt to do so will result in a visual warning message.

D (Return) - This command results in the updated version of the propeller palm base being plotted. When plotting is complete, a single bell sounds signifying control is at the primary invisible command status.

B (Return) - This command causes a title to be printed and a border to be drawn.

E (Return) - This command permits the user to select the elements for removal and replacement by bolt and bolthole substructures. When a cursor appears, the user may begin identifying elements for removal. Starting and ending on any corner node, proceed around an element in a clockwise direction identifying all element nodes including midside nodes. Nodal identification is achieved by locating the cursor at the node and pressing any alphanumeric key except "T and "S". An "x" will then be drawn at the node.

To aid the user in element removal, a number will appear in the top left hand corner of the screen stating which node of the current edge is to be identified next. After the third node of each edge on the element is identified, the program prompts the user for the number of elements along the edge. If the current element edge forms part of the bolthole superelement boundary, the number supplied will be the number of elements employed along the same edge in the bolthole substructure. Typically one or two elements generally prove adequate but for high accuracy greater refinement in the substructure may be desirable. Arbitrary numbers may be supplied if the current element edge does not form part of the bolthole substructure boundary.

If a node selection error is made, an "E" can be entered and the number at the screen decremented so that node selection can be again attempted. This feature can be repeated but the node counter should not decrease below 1 on the first edge and below 2 on subsequent edges.

After all elements that will be replaced by one bolt and bolthole substructure pair have been selected, the user must enter a "T". The user may then proceed to select more elements for the next substructure pair or enter an "S" to end the element selection procedure. Program control remains at the primary level and any of the invisible commands may be activated. For instance the "D" prompt would now produce a plot of the palm minus all elements identified for removal.

S (Return) - This command terminates modifications of the propeller palm with the terminal operated in the invisible command mode. Data tapes for subsequent analysis are created. Simultaneously data integrity checks are performed to identify any model deficiencies which might exist. The following visual terminal responses can now be expected.

(3) TYPE 1 TO CONTINUE MODEL MODIFICATION;
0 TO END MODEL MODIFICATION.

Enter 1 or 0 in free format according to whether model deficiencies in the form of bolthole substructure not formed at bolt locations. Entering 1 returns control to the invisible command mode where further node relocation, element selection for bolt and bolthole replacement, etc. can be performed. Entering 0, on the other hand, results in program execution proceeding to the next subprogram step.

(4) ENTER 1 TO RETAIN ORIGINAL MODEL;
0 TO PROCESS MODIFICATIONS.

If the user wishes to retain the original grid coordinates and element connections rather than replace them with the modified model, enter a non-zero number in free format. This will be desirable for instance if through grid modification two nodes are inadvertently given the same position since their subsequent separation would probably prove impossible without time-consuming manual edits of program data. Restarting mesh modifications with the original grid will probably prove easier.

(5) ENTER 1 TO SELECT NODES FOR GAP ELEMENTS AND/OR BOUNDARY CONDITIONS, OTHERWISE ENTER 0.

This option is used to select nodes on the propeller palm base which are connected to ground via gap elements (i.e. elements offering no resistance to tension) or are to have boundary conditions assigned to them. When the user response is 0, subprogram execution is ended.

(6) (The base of the propeller palm viewed from above is displayed on the screen and the nodes marked.)

(7) ENTER 1 TO WINDOW A PORTION OF THIS PLOT, OTHERWISE ENTER 0.

The windowing feature is useful when choosing gap elements or assigning nodal boundary conditions if the grid is fine. It permits magnification of a small portion of the original plot, where either node locations or node numbering are unclear because of the proximity of nodes.

When this feature is activated the terminal responds with two bells and the cursor appears. The user should first position the cursor in the lower left hand corner of the desired window and press any alphanumeric keyboard character except "S" and then repeat this procedure for the top right hand corner. At this point a box with sides parallel to the screen axes is drawn to enclose the area selected for magnification.

The user should then press the return button to have the screen cleared and the windowed portion redrawn to fill the whole screen.

(8) ENTER 1 FOR KEYBOARD INPUT;
2 FOR INPUT VIA THE GRAPHICS CURSOR.

If 1 is entered, the nodes will be labelled to assist the user in node selection; if 2 is chosen, the graphics cursor appears. In addition all gap elements or boundary conditions defined in earlier runs are shown on the drawing. An "x" is used to designate a gap element whereas a "+" is used for a boundary condition.

The user may then commence selecting nodes using the method selected above. For the keyboard input method, the prompt ">" will be repeatedly issued and new gap elements or boundary conditions can be entered. To delete a gap element or boundary condition enter the negative value of the node. To end the selection cycle enter 0.

For the cursor selection method, move the cursor to desired node and press any key except "D" or "S". Entering "D" will delete the gap element or boundary condition (upon which a square will be drawn at the node) and "S" terminate the selection cycle.

(9) ENTER 1 TO CONTINUE SELECTING NODES;
0 TO STOP.

Entering 1 will allow the node selection procedure to be repeated through transfer of control to (5). This is desirable when node selection is accomplished with windowing and only portions of the propeller base can be examined at any one time.

(10) GAP ELEMENT AND BOUNDARY CONDITION DATA APPENDED TO END OF GEOMETRY FILE PREFX.PLM.

B.8C PVBLTC: Plotting the base of bolthole substructures for identification of nodes for gap elements and boundary conditions.

Terminal Response:

(1) SUBPROGRAM PVBLTC PROGRAM TO PLOT THE BASE OF THE BOLTHOLE SUBSTRUCTURE FOR IDENTIFICATION OF NODES FOR GAP ELEMENTS AND/OR BOUNDARY CONDITIONS.

(2) BOLTHOLE SUBSTRUCTURE #XXX

ENTER 1 TO SELECT NODES FOR GAP ELEMENTS AND/OR BOUNDARY CONDITIONS, OTHERWISE ENTER 0.

This option is used to select nodes on the base of the bolthole superelements within the propeller palm which are to be connected to ground via gap elements (i.e. elements offering no resistance to tension) or are to have boundary conditions assigned to them. When the user response is 0, subprogram execution is terminated.

(3) (Base of the current bolthole substructure viewed from above is displayed with all nodes marked.)

(4) ENTER 1 TO WINDOW A PORTION OF THIS PLOT, OTHERWISE ENTER 0.

The windowing feature is useful when choosing gap elements or assigning nodal boundary conditions if the grid is fine. It permits magnification of a small portion of the original plot, where either node locations or node numbering are unclear because of the proximity of nodes.

When this feature is activated the terminal responds with two bells and the cursor appears. The user should first position the cursor in the lower left hand corner of the desired window and press any alphanumeric keyboard character except "S" and then repeat this procedure for the top right hand corner. At this point a box with sides parallel to the screen axes is drawn to enclose the area selected for magnification.

The user should then press the return button to have the screen cleared and the windowed portion redrawn to fill the whole screen.

(5) ENTER 1 FOR KEYBOARD INPUT;
2 FOR INPUT VIA THE GRAPHICS CURSOR.

If 1 is entered, the nodes will be labelled to assist the user in node selection; if 2 is chosen, the graphics cursor appears. In addition all gap elements or boundary conditions defined in earlier runs are shown on the drawing. An "x" is used to designate a gap element whereas a "+" is used for a boundary condition.

The user may then commence selecting nodes using the method selected above. For the keyboard input method, the prompt ">" will be repeatedly issued and new gap elements or boundary conditions can be entered. To delete a gap element or boundary condition enter the negative value of the node. To end the selection cycle enter 0.

For the cursor selection method, move the cursor to desired node and press any key except "D" or "S". Entering "D" will delete the gap element or boundary condition (upon which a square will be drawn at the node) and "S" terminate the selection cycle.

(6) ENTER 1 TO CONTINUE SELECTING NODES FOR CURRENT BOLTHOLE SUPERELEMENT;
0 TO PROCEED TO NEXT BOLTHOLE SUPERELEMENT.

Entering 1 permits the node selection procedure to be continued through transfer of control to (4). This is desirable when node selection is accomplished with windowing and only portions of the propeller base can be examined at any one time.

Entering 0, permits the node selection procedure for identification of model gap elements and boundary conditions to be carried out for the next bolthole substructure with a transfer of control to (2). If all bolthole substructures have been considered, control transfers to (7).

(7) FILE PREFX.BHO HAS BEEN UPDATED.

Because master nodes are required at nodes connected to ground via gap elements or with boundary conditions assigned, the master node list on file is suitably updated as are lists of gap elements and boundary conditions.

APPENDIX C

GENERATION OF LOADS

C.1 PVLODA: Adjusts applied propeller pressure loading to reduce pressure spikes along the leading and trailing edges from which loading errors develop.

Note: Input load data to be supplied on file PREFX.LIN (see following Section C.2 for format).

Terminal Response:

(1) ENTER 0 TO NOT ADJUST THE LOAD
1 TO ADJUST LEADING EDGE ONLY
2 TO ADJUST BOTH LEADING AND TRAILING EDGES

(2) HOW DO YOU WISH TO ADJUST THE MAXIMUM PEAK PRESSURES
0 = REDUCE PEAK VALUES TO MAXIMUM LIMITS ONLY
1 = ADJUST WITH NO LIMITS
2 = ADJUST USING MAXIMUM LIMITS

Peak pressures on the leading and/or trailing edge regions of the blade can be reduced to maximum levels as specified by the user in the following responses. Adjustment of the pressure data over the leading edge and trailing edge finite elements can also be done automatically by this program in such a way to maintain the thrust and torques. Note, the first option of cutting the peak levels may introduce errors in the thrust and torques but is a useful step such that in VAST the quadratic fit for the pressure over the finite elements does not produce large undershoots.

(3) Omitted if 1 was entered in (2)

(a) ENTER MAXIMUM PEAK PRESSURE AT LEADING EDGE

(b) Omitted if 1 was entered in (1)

ENTER MAXIMUM PEAK PRESSURE AT TRAILING EDGE

(4) Omitted if 0 was entered in (2)

(a) INPUT THE FRACTIONAL CHORDWISE LOCATIONS OF THE CORNER NODES OF THE LEADING EDGE FINITE ELEMENT

(b) Omitted if 1 was entered in (1)

INPUT THE FRACTIONAL CHORDWISE LOCATIONS OF THE CORNER NODES OF THE TRAILING EDGE FINITE ELEMENT

(5) Omitted for static pressure loading

(a) THERE ARE (NTIME) TIME POINTS OF DYNAMIC LOAD (ASSUMES ONE CYCLE OF DATA)

(b) FREQUENCY OF THIS PRESSURE LOAD IS ____cps

C.2 PVLODB: Transforms static or dynamic propeller blade pressure data into form acceptable to PVLODV.

Note: When the program is operated in the overlay mode (i.e. MODE=2 on page 5.1-1), load data is to be supplied on file PREFX.LIN or LAD. In the standalone operational mode (i.e. MODE=1), source of data is determined by the first data card.

Header Card (A6) (Omit for MODE=2)

LOADB (See Appendix K for further details on use of header cards.)

Card 1 (2I5) (Omit for MODE=2)

IDATA = Code defining pressure load data source.
= 1, read Cards 2-13 from file PREFX.CDR.
= 3, read Cards 2-13 from file PREFX.LIN or LAD (formatted).

IPTC = 0, printout is reduced.

Card 2 (I5)

NDATA = Number of blade loading data sets.

Repeat Cards 3 to 13 for each data set.

Card 3 (3I5)

IPRESS = Code for specifying type of pressure loading. If = 1, a uniform pressure loading is generated using a specified face pressure and a specified back pressure. If = 2, pressures at specified radial and chordwise locations are provided, thus permitting non-uniform pressure loading.

ILTYPE = Code describing the type of loading. If = 1, the loading is considered to be static. If = 2, the loading is considered to be dynamic. If = 3, the dynamic loading is considered to be steady periodic. The user describes the loading function for one complete period including end points. The desired number of repeating periods will be generated by PVLODV.

the loading function for one complete period including end points. The desired number of repeating periods will be generated by PVLODV.

ILPER = Total number of periods for dynamic loading function. Set zero if ILTYPE $\neq$ 3.

Card 4 (2I5) (Omit if IPRESS=1)

K = Number of radial positions where pressures are specified (≥ 2).

L = Number of chordwise positions where pressures are specified (≥ 2).

Card 5 (8F10.4) (Omit if IPRESS=1)

$R_{(1)}$
↓ = The fraction radii where pressures are specified.
$R_{(K)}$

Card 6 (8F10.4) (Omit if IPRESS=1)

$C_{(1)}$ = The fraction chords where pressures are specified.
↓
$C_{(L)}$

Card 7 (I5) (Omit if ILTYPE=1)

NLV = Number of points in time; i.e. the number of pressure values used to describe dynamic load history.

Provide Cards 8 to 11 for NLV points in time.

Card 8 (E10.3) (Omit if ILTYPE=1)

TIME = Time in seconds.

Card 9 (2F10.4) (Omit if IPRESS=2)

UPB = Uniform pressure on back (force/length^2). See Figure C.1(a). (Positive towards surface.)

UPF = Uniform pressure on face (force/length^2). See Figure C.1(a). (Positive towards surface.)

Repeat Cards 10 and 11 for i=1,K

Card 10 (PB_{ij}, j=1,L) (8F10.4) (Omit if IPRESS=1)

$PB_{(i,1)}$ = Pressure loading on back (force/length^2). See Figure C.1(b). (Positive towards surface.)
↓
$PB_{(i,L)}$

Card 11 (PF_{ij}, j=1,L) (8F10.4) (Omit if IPRESS=1)

$PF_{(i,1)}$ = Pressure loading on face (force/length^2). See Figure C.1(b).
↓
$PF_{(i,L)}$

Card 12 (I5)

NDF = Number of radial positions where lumped drag forces at mid- chord are specified.

Card 13 (2F10.4) (Omit if NDF=0)

RD_i = The fraction radii where drag forces are specified.

DF_i = Drag force.

Repeat Card 13 for i=1,NDF
Repeat Card 13 for NLV points in time

C.3 **PVLODV:** Generates load data for the PVAST finite element model of the propeller blade.

Header Card (A6)

LOADV (See Appendix K for further details on use of header cards.)

Card 1 (5I5)

ICF ≠ 0 , loads due to centrifugal forces will be generated. Must = 0 for dynamic loading analysis by VAST. (Set to zero when subprogram PVLODV is executed in the overlay mode, i.e. MODE=2 as per page 5.1)

IPL ≠ 0, loads due to pressure will be generated. (Set to zero for MODE = 2.)

IGRID = as defined in PVBLD2 data (Appendix B.3).

IBASE = as defined in PVBLD2 data (Appendix B.3).

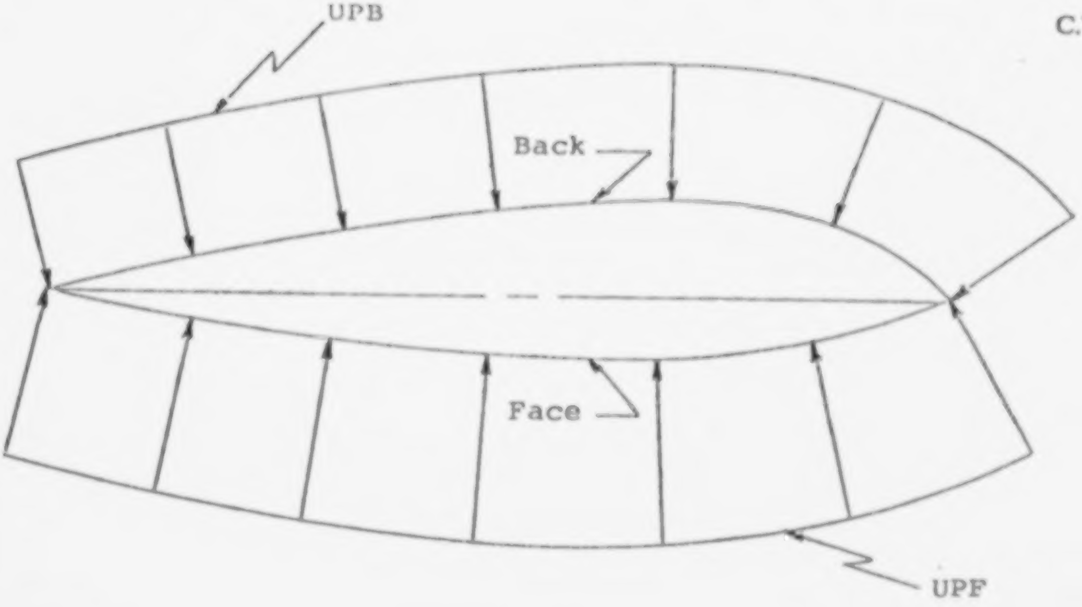
IELEM = as defined in PVBLD2 data (Appendix B.3).

ISUB ≠ 0, blade modelled as a superelement and bottom element row replaced by transition elements if IELEM=0.

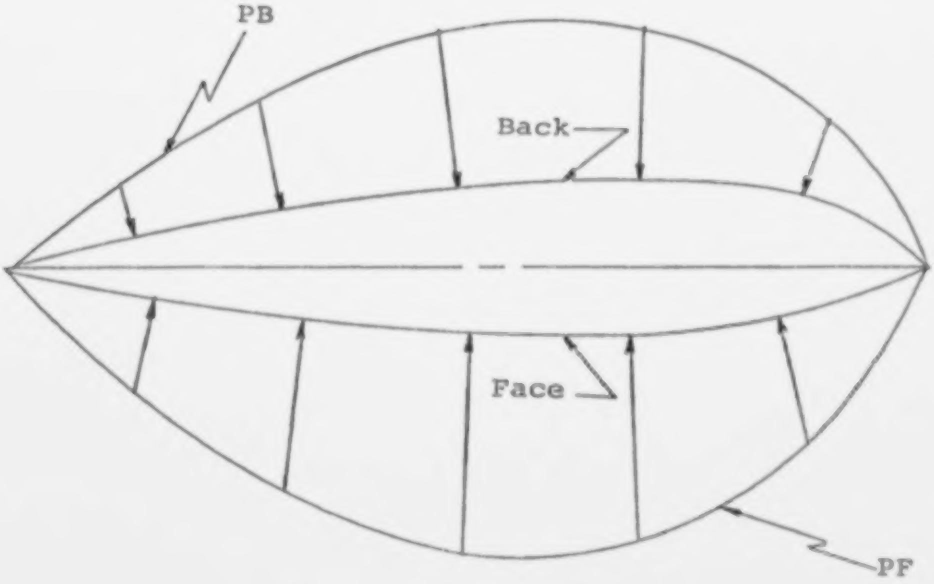
Card 2 (2E10.3) (Omit if ICF=0 and subprogram PVLODV is executed in the stand alone mode, i.e. MODE = 1)

ANGVZ = Rotational velocity (cycles/sec)

DEN = Material weight per unit volume/g (force-time^2/length^4).


UPB
Back
Face
UPF

(a) Uniform Pressure


PB
Back
Face
PF

(b) Non-uniform Pressure

FIGURE C.1: Pressure Loading On Blade

APPENDIX D

INPUT DATA FOR POST-PROCESSING PROGRAMS

D.1 PVDS1: Processes results from static analysis for subsequent plotting by PGDIS2/3 and PGSTR1/2/3.

Card 1 (I5)

IPTC = Printing code. If IPTC ≠ 0, displacements and stresses are printed. If IPTC=0, this printout is suppressed.

D.2 PVDS2: Processes results from dynamic analysis for subsequent plotting by PGDS1.

Card 1 (2I5)

IOPT = 1, dynamic displacement and stress histories are computed.
= 2, dynamic displacement histories only are computed.

IPTC = Printing code. If IPTC $\neq$ 0, dynamic displacement and stress histories are printed. If IPTC=0, this printout is suppressed. (Note: maximums are printed regardless of the value of IPTC).

D.3 PVFOR1: Processes force vector data for plotting by PGFOR1.

Card 1 IPTC (I5)

IPTC = Printing code. If IPTC ≠ 0, displacements and stresses are printed. If IPTC=0, this printout is suppressed.

D.4 PVSECT: Interactive PVAST post processing program for the identification of element surfaces on either horizontal or vertical propeller cross-sections for stress contouring purposes. PVSECT identifies vertical surfaces in the root (if it exists) as well as the blade but horizontal surfaces in the blade only. File created serves as input to POSTV2.

Terminal Response:

(1) ENTER 5 CHARACTER FILE NAME PREFIX:

Supply the unique VAST file name prefix used for the current propeller analysis.

(2) ENTER ELEMENT SURFACE DATA FILE NAME PREFIX:

The prefix name assigned to the file generated with element surface data for the desired cross-section.

(3) IS THIS A SUPERELEMENT ANALYSIS? (0=NO)

If no, the root model is presumed to not exist.

(4) ENTER IBASE (REFER TO PVBLD2 FOR DESCRIPTION)

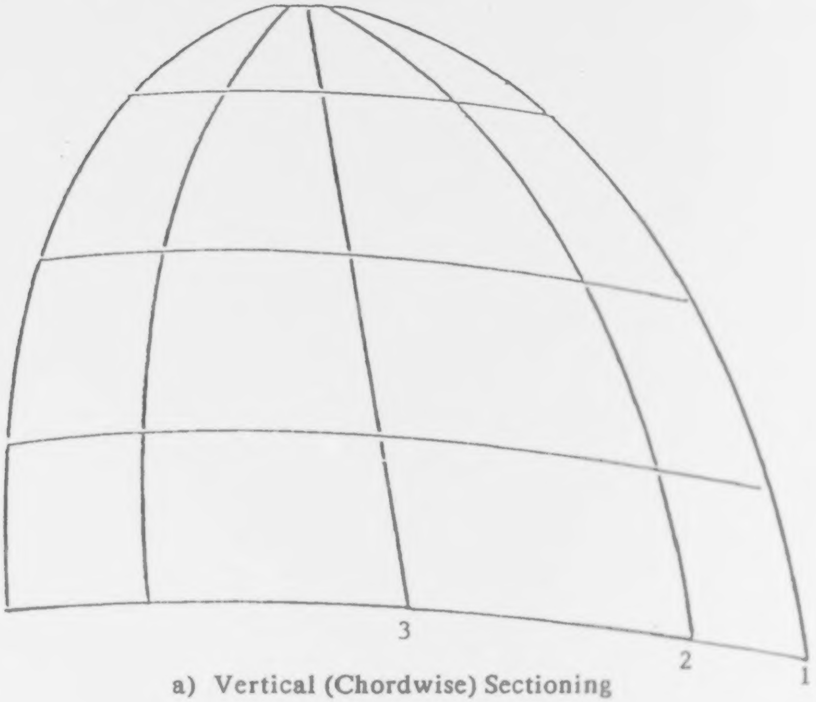
This response will not appear if the response to (3)is not 0.

(5) ENTER 1 FOR A VERTICAL CROSS-SECTION
2 FOR A HORIZONTAL CROSS-SECTION

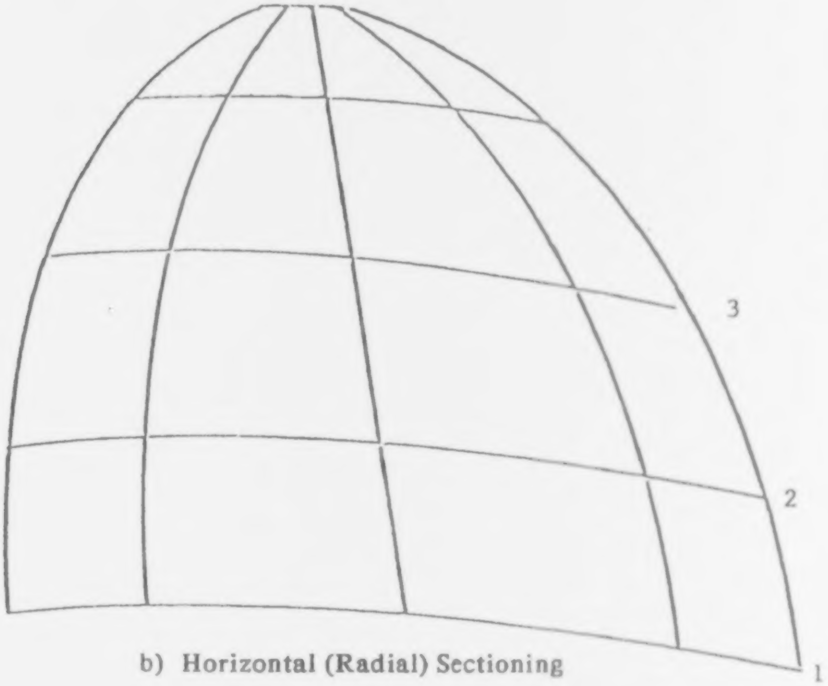
(6) ENTER CHORDWISE SECTION NUMBER
This prompt appears if the response to (5) is 1.

(7) ENTER RADIAL SECTION NUMBER
This prompt appears only if the response to (5) is 2.

NOTE: See Figure D.1 for a description of the cross-section numbers



a) Vertical (Chordwise) Sectioning



b) Horizontal (Radial) Sectioning

FIGURE D.1: Section Definition for Specifying Element Surface

D.5 PVFOR2: Computes from the results of a PVAST dynamic analysis for a single propeller blade (operating in a wake), the time histories of force and moment resultants acting on the shaft for a single blade or for the whole (multi-bladed) propeller.

Card 1 (I5)

INCPR = Parameter for reducing output. Forces and moments are printed out after every INCPR time interval

Card 2 (I5, E10.3)

NBLAD = Number of blades in assembled propeller

RPM = Propeller speed (revolutions/minute). (Note: If a frequency response analysis was performed, the forcing frequency will take precedence over this value)

Card 3 (4E10.3)

HWT = Weight of propeller hub/(density*g)

HIX, HIY, HIZ = Mass moments of inertia of hub with respect to the x, y and z axes/(density*g)

Note: Weight is in units of force (or mass-length-sec^{-2})
Density is in units of (mass-length^{-3})
Mass moment of inertia is in units of (mass-length^{2})
g (gravitational acceleration is in units of (length-sec^{-2})

APPENDIX E

OPERATING INSTRUCTIONS FOR PLOTTING PROGRAMS

E.1 PGBLAD: Plots expanded outlines, developed sections, developed outlines, transverse projections, longitudinal projections, plan views and grid proportions for the blade finite element model.

Terminal Response:

(1) SUBPROGRAM PGBLAD PROPELLER GRAPHICS PROGRAM FOR PLOTTING EXPANDED OUTLINES, DEVELOPED SECTIONS, DEVELOPED OUTLINES, TRANSVERSE PROJECTIONS, LONGITUDINAL PROJECTIONS, PLAN VIEWS AND GRID PROPORTIONS FOR THE BLADE FINITE ELEMENT MODEL.

PLOT NUMBER	DEFINITION
1	F.E. EXPANDED OUTLINE.
2	F.E. DEVELOPED SECTIONS.
3	F.E. DEVELOPED OUTLINE.
4	F.E. TRANSVERSE PROJECTION.
5	F.E. LONGITUDINAL PROJECTION.
6	F.E. PLAN VIEW.
7	F.E. GRID PROPORTIONS.

SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).

Enter number of desired plot, and a reducing factor as a percent, which is used to reduce the size of the plot. These are integers and must be entered in free-format.

(2) (If plot number 7 has been requested.)

SPECIFY ROTATIONS ABOUT Z, Y, AND X AXES

Enter, in degrees, the desired orientation of the plot. These are real numbers and must be entered in free format. When zeros are specified for all three axes, the global X and Y axes correspond to the screen's horizontal and vertical axes, respectively; the global Z axis is out of the screen.

(3) (Plot appears on screen)

When plot is completed, enter a character and press return key (this may have to be repeated).

(4)
```
TYPE   1 TO CONTINUE;
       0 TO END THIS PLOTTING OPTION.
```

Enter 1 or 0 in free format. If 1 is entered, another plot can be obtained. If 0 is entered, execution is terminated.

E.2A PGLODI: Plots input and adjusted pressure loading chordwise distributions as well as time histories for dynamic pressure loading.

Terminal Response:

(1) (a) (Omitted if load is dynamic)

```
LOAD DATA IS STATIC
ENTER F FOR FACE PRESSURES
ENTER B FOR BACK PRESSURES
```

go to response (2)

(b) (for dynamic loading)

```
THERE ARE (NTIME) TIME POINTS USED TO DEFINE DYNAMIC
PRESSURE LOAD TIME HISTORY.

ENTER T TO PLOT CHORDWISE DISTANCE AT A SELECTED TIME
POINT;
ENTER P TO PLOT THE HISTORY AT A SELECTED LOCATION ON THE
BLADE;
ENTER E TO END THIS PLOTTING OPTION.
```

If E is entered, PVAST4 continues with processing.

(c) (If T is entered in (b))

```
THERE ARE (NTIME) TIME POINTS.
WHICH TIME POINT IS TO BE PLOTTED
```

(d)

```
ENTER F FOR FACE PRESSURES
ENTER B FOR BACK PRESSURES
```

go to response (2)

(e) (If P is entered in (b))

```
THERE ARE (K) RADIAL LOCATIONS FOR THIS PRESSURE SET
THEY ARE:
```

List of fractional radial locations are written

```
SELECT THE FRACTION RADIUS OF THE LOCATION TO BE PLOTTED
```

(f) THERE ARE (L) CHORDWISE LOCATIONS FOR THIS PRESSURE SET THEY ARE:

List of fractional chordwise locations are written

SELECT THE FRACTION CHORD OF THE LOCATION TO BE PLOTTED

Plot of dynamic pressure time history appears

(g) SELECT ANOTHER LOCATION (0=NO, 1=YES)

If 1 is entered, (e) appears
If 0 is entered, (b) appears

(2) SELECT THE LOAD CURVE TO BE PLOTTED
ENTER THE NUMBER OF THE RADIAL SECTION FOR THE INDIVIDUAL PLOT, (1 to K)
OR ENTER (K+1) FOR ALL LOAD CURVES

Note: K is the number of radial positions where pressure distributions were specified in the input data. If a number 1 to K is entered, individual chordwise pressure distributed is plotted. If a number greater than K is entered, all pressure distribution are plotted on the same plot.

(3) (a) (After each plot)

ENTER F FOR FACE PRESSURES
ENTER B FOR BACK PRESSURES
ENTER R FOR CHANGE TO ANOTHER RADIUS
ENTER E TO END PLOTTING

If F,B or R is entered, (2) reappears.

(b) (For dynamic load, if E is entered)

SELECT ANOTHER TIME POINT (0=NO, 1=YES)

If 1 is entered, 1(c) appears.
If 0 is entered, 1(b) appears.

E.2B PGLOD2: Plots static or dynamic pressure loadings.

Terminal Response:

(1) THERE ARE XX LOADING FUNCTIONS:
ENTER DESIRED LOADING FUNCTION:

Enter number representing loading function of interest in free format. This response will not appear if there is only 1 loading function.

(2)

PLOT NUMBER	DEFINITION
1	BASIC CHORDWISE PRESSURE LOAD DATA
2	COMPARISON OF BASIC CHORDWISE PRESSURE LOADS AND FINITE ELEMENT DATA
3	FINITE ELEMENT PRESSURE LOADS

SURFACE NUMBER	DEFINITION
1	FRONT (PRESSURE) SURFACE
2	BACK (SUCTION) SURFACE

SPECIFY PLOT NUMBER AND SURFACE NUMBER:

Enter plotting options in free format.

(3) SPECIFY PRESSURE UNITS TO APPEAR ON PLOT:

(FOR UNITS:	NIL	PSI	KSI	PSF	KSF	PA	KPA	MPA
ENTER:	0	1	2	3	4	5	6	7)

Select number associated with units of interest and enter in free format.

(4) (For dynamic pressure loading only)

TIME(SEC)=XXX
TYPE 1 TO PLOT, OR 0 TO SKIP

Enter 1 or 0 in free format. If 1 is entered, pressure loading at TIME XXX seconds is plotted. If 0 is entered, plotting is suppressed.

(5) (For dynamic analysis only)

NO MORE TIME POINTS; BEGIN AGAIN AT TIME POINT #1? (0=NO)

This response appears if the user requests another time point after the last time point has been presented. Enter a non-zero number, in free format to continue plotting; otherwise enter 0.

(6) (Only if plot number option 2 was selected)

FINITE ELEMENT FRACTION RADII

X.XXXX X.XXXX X.XXXX etc.

SPECIFY FRACTION RADIUS FOR PLOT OF PRESSURE CURVES

Enter one radius from above list in free format.

(7) (Plot appears on the screen)

A hard copy may be made at this time. To continue, the user must press the "Return" key.

(8) WINDOWING OPTION? (0=NO)

If the terminal has cursor capability, the user may use it to window the curve labels in plotting options 2 and 3 resulting in the windowed portion being "magnified" and thus making the labels easier to read. To window, enter a number other than 0 in free format. A cursor will then appear. Locate the lower left hand corner of the desired window and press any alphanumeric keyboard character. Then locate the top right hand corner and again press any key. The screen will then clear and the magnified plot of the windowed area will appear. To continue, press the "Return" key.

(9) (Only if plot number option 2 was selected)

ENTER 1 FOR A PLOT OF ANOTHER RADIUS ON THE SAME SURFACE OTHERWISE ENTER 0:

Enter 1 or 0 in free format. If 1 is entered, the user may plot another radius; if 0 is entered, the following response appears.

(10) (For dynamic analysis only)

TYPE 1 FOR ANOTHER TIME POINT. OTHERWISE ENTER 0.

If 1 is entered control is transferred to either (4) or (5). Otherwise, the following prompt appears.

(11) ENTER 0 TO END THIS PLOTTING OPTION;
OR 1 TO SPECIFY A NEW PLOT NUMBER;
OR 2 TO SPECIFY A NEW LOADING FUNCTION.

Respond in free format. If 0, program PVAST4 continues; if 1, plotting options (2) are again displayed; if 2, a new loading function (1) may be selected. Option number 2 will not appear if there is only 1 loading function.

E.3 PGFEM: Plots finite element grids for blade and palm/hub models.

Terminal Response:

(1) SUBPROGRAM PGFEM PROPELLER GRAPHICS PROGRAM FOR PLOTTING THE FINITE ELEMENT GRID OR PROPELLER BLADE AND ROOT MODELS.

SPECIFY PLOT REDUCING FACTOR (PERCENT).

Enter plot reducing factor in free format.

(2) SPECIFY ROTATIONS ABOUT Z, Y, AND X AXES

Enter, in degrees, the desired orientation of the plot. These are real numbers and must be entered in free format. When zeros are specified for all three axes, the global X and Y axes correspond to the screen's horizontal and vertical axes, respectively; the global Z axis is out of the screen.

(3) SPECIFY FIRST AND LAST ELEMENT TO BE PLOTTED.

Enter 2 integers in free format.

(4) TYPE 1 TO HAVE ORIGIN LOCATED.

(5) AFTER PLOTTING USE:

S - TO SAVE PLOT TO PREFIX.L00 FILE
PRESS "RETURN" TO PLOT

Press the return key to plot the finite element model.

(6) Plot appears on screen. If it is desired to save the plot type S and press return. The plot is saved to a PREFIX.L** file, where ** starts from 00 and incremented by 1 each time this option is implemented in the same session.

(7) ENTER 1 TO CONTINUE; 0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.4 PGDIS1: Plots deflected blade shapes resulting from static loading and/or natural mode shapes from a free vibration analysis.

Terminal Response:

(1) SUBPROGRAM PGDIS1 PROPELLER GRAPHICS PROGRAM FOR PLOTTING DISTORTED BLADE SHAPE RESULTING FROM STATIC LOADING AND/OR NATURAL MODE SHAPES FROM A FREE VIBRATION ANALYSIS.

PLOT NUMBER	DEFINITION
1	FINITE ELEMENT GRID.
2	DISPLACEMENTS.
3	MODE SHAPES.

SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).

Enter plot number and reducing factor in free format.

(2) SPECIFY ROTATIONS ABOUT Z, Y, AND X AXES

Enter, in degrees, the desired orientation of the plot. These are real numbers and must be entered in free format. When zeros are specified for all three axes, the global X and Y axes correspond to the screen's horizontal and vertical axes, respectively; the global Z axis is out of the screen.

(3) (If plot number 2 has been requested):

SPECIFY SCALING FACTOR FOR DISPLACEMENTS.

Enter number in free format.

(4) (If plot number 3 has been requested):

SPECIFY SCALING FACTOR FOR NATURAL MODE SHAPES.

Enter number in free format.

E.5 PGDS1: Provides displacement and stress time histories.

Terminal Response:

(1) SUBPROGRAM PGDS1 PLOTS DISPLACEMENT AND STRESS TIME HISTORIES.

THIS PROGRAM PROVIDES THE FOLLOWING TIME HISTORY PLOTS AT SPECIFIED RADIAL AND CHORDWISE POSITIONS:

PLOT NUMBER	DEFINITION
1	DISPLACEMENT IN GLOBAL X DIRECTION.
2	DISPLACEMENT IN GLOBAL Y DIRECTION.
3	DISPLACEMENT IN GLOBAL Z DIRECTION.
4	NORMAL DISPLACEMENT COMPONENT.
5	TANGENTIAL DISPLACEMENT COMPONENT.
6	RADIAL DISPLACEMENT COMPONENT.
7	PRIMARY PRINCIPAL STRESS.
8	SECONDARY PRINCIPAL STRESS.
9	TANGENTIAL STRESS COMPONENT.
10	RADIAL STRESS COMPONENT.

SPECIFY PLOT NUMBER.

Enter integer in free format.

(2) RADIAL POSITIONS (FRACTION).

X.XXX X.XXX X.XXX etc.

SPECIFY RADIAL POSITION.

Enter real number in free format.

(3) CHORD POSITIONS (FRACTION).

X.XXX X.XXX X.XXX etc.

SPECIFY CHORD POSITION.

Enter real number in free format.

(4) Plot appears on screen.

(5) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.6 PGSTR1: Plots stresses for static analysis.

Terminal Response:

(1) PROGRAM PGSTR1 PROPELLER GRAPHICS PROGRAM FOR PLOTTING BLADE STATIC STRESSES.

THIS PROGRAM PROVIDES THE FOLLOWING PLOTS:

PLOT NUMBER	DEFINITION
1	CHORDWISE DISTRIBUTION OF PRIMARY PRINCIPAL STRESSES.
2	CHORDWISE DISTRIBUTION OF SECONDARY PRINCIPAL STRESSES.
3	CHORDWISE DISTRIBUTION OF TANGENTIAL STRESS COMPONENT.
4	CHORDWISE DISTRIBUTION OF RADIAL STRESS COMPONENT.

SPECIFY PLOT NUMBER.

Enter integer in free format.

(2) RADIAL POSITIONS (FRACTION).

X.XXX X.XXX X.XXX etc.

SPECIFY RADIAL POSITION.

Enter real number in free format.

(3) Plot appears on screen.

(4) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.7 PGDIS2: Plots displacements for static analysis.

Terminal Response:

(1) SUBPROGRAM PGDIS2 PROPELLER GRAPHICS PROGRAM FOR PLOTTING BLADE STATIC DISPLACEMENTS.

THIS PROGRAM PROVIDES THE FOLLOWING PLOTS:

PLOT NUMBER	DEFINITION
1	CHORDWISE VARIATION OF NORMAL DISPLACEMENT COMPONENT.
2	CHORDWISE VARIATION OF TANGENTIAL DISPLACEMENT COMPONENT.
3	CHORDWISE VARIATION OF RADIAL DISPLACEMENT COMPONENT.

SPECIFY PLOT NUMBER.

Enter integer in free format.

(2) RADIAL POSITIONS (FRACTION).

X.XXX X.XXX X.XXX etc.

SPECIFY RADIAL POSITION.

Enter real number in free format.

(3) Plot appears on screen.

(4) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.8 PGDIS3: Plots loaded blade sections superimposed on unloaded sections.

Terminal Response:

(1) SUBPROGRAM PGDIS3 PROPELLER GRAPHICS PROGRAM FOR PLOTTING LOADED BLADE SECTIONS SUPERIMPOSED ON UNLOADED SECTIONS.

THIS PROGRAM PROVIDES THE FULLY-DEVELOPED VIEWS OF BLADE SECTIONS UNDER LOAD, AND SUPERIMPOSED ON THE UNLOADED SECTIONS TO SHOW SECTION DISTORTION.

RADIAL POSITIONS (FRACTION).

X.XXX X.XXX X.XXX etc.

SPECIFY RADIAL POSITION AND SCALE FACTOR.

Enter two numbers in free format.

(2) Plot appears on screen.

(3) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.9 PGSTR2: Plots principal stress contours for static analysis.

Terminal Response:

(1)
```
SUBPROGRAM PGSTR2   PROPELLER GRAPHICS PROGRAM FOR PLOTTING
                    PRINCIPAL STRESS CONTOURS (STATIC ANALYSIS
                    ONLY).

THIS PROGRAM PROVIDES THE FOLLOWING PLOTS:

PLOT NUMBER              DEFINITION

     1                   PRINCIPAL STRESS CONTOURS (FACE).
     2                   PRINCIPAL STRESS CONTOURS (BACK).

SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).
```

Enter plot number and reducing factor in free format.

(2)
```
PRIMARY OR SECONDARY (TYPE 1 OR 2).
```

Enter 1 or 2 in free format.

(3)
```
STRESS RANGE = ______. SPECIFY CONTOUR INTERVAL.
```

Enter number in free format.

(4) Plot appears on screen and cursor is activated.

Move cursor to location where contour label is wanted and enter any symbol except S from the keyboard (S terminates the contour labelling).

(5)
```
TYPE  1 TO CONTINUE;
      0 TO END THIS PLOTTING OPTION:
```

Enter 1 or 0 in free format.

E.10 PGDIS4: Plots natural mode shapes superimposed on unloaded blade section.

Terminal Response:

(1)

```
SUBPROGRAM PGDIS4   PROPELLER GRAPHICS PROGRAM FOR PLOTTING
                    NATURAL MODE SHAPES FROM A FREE VIBRA-
                    TION ANALYSIS SUPERIMPOSED ON THE
                    UNLOADED BLADE SECTIONS.

THIS PROGRAM PROVIDES THE FULLY-DEVELOPED VIEWS OF BLADE
SECTIONS UNDER LOAD, AND SUPERIMPOSED ON THE UNLOADED
SECTIONS TO SHOW SECTION DISTORTION.

RADIAL POSITIONS (FRACTION).

X.XXX  X.XXX  X.XXX  etc.

SPECIFY RADIAL POSITION AND SCALE FACTOR (PERCENT).
```

Enter two numbers in free format.

(2) Plot appears on screen.

(3)

```
TYPE 1 TO CONTINUE;
     0 TO END THIS PLOTTING OPTION:
```

Enter 1 or 0 in free format.

E.11 PGDIS5: Plots contour lines of mode shapes.

Terminal Response:

(1)

```
SUBPROGRAM PGDIS5     PROPELLER GRAPHICS PROGRAM FOR PLOTTING
                      CONTOUR LINES OF MODE SHAPES.

THIS PROGRAM PLOTS CONTOUR LINES OF MODE SHAPES ON TWO BLADE
VIEWS.
```

PLOT NUMBER	DEFINITION
1	TRANSVERSE.
2	LONGITUDINAL.

```
SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).
```

Enter plot number and reducing factor in free format.

(2)

```
SPECIFY MODE NUMBER.
```

Enter mode number in free format.

(3)

```
SPECIFY CONTOUR INTERVAL (0.1 TO 1.0).
```

Enter contour interval in free format.

(4)

```
PROJECTED GLOBAL OR NORMAL DISPLACEMENTS (1 OR 2).
```

Enter 1 or 2 in free format.

(5) Plot appears on screen.

(6)

```
TO PLOT ANOTHER MODE WITH SAME OPTIONS, ENTER (NON-ZERO) MODE
NUMBER:
```

Enter mode number in free format.

(7)

```
TYPE   1 TO CONTINUE;
       0 TO END THIS PLOTTING OPTION:
```

Enter 1 or 0 in free format.

E.12 PGSTR3: Plots principal stress vectors for static analysis.

Terminal Response:

(1) SUBPROGRAM PGSTR3 PROPELLER GRAPHICS PROGRAM FOR PLOTTING PRINCIPAL STRESS VECTORS (STATIC ANALYSIS ONLY).

Program for plotting principal stress vectors.

PLOT NUMBER	DEFINITION
1	PRINCIPAL STRESS VECTORS (FACE).
2	PRINCIPAL STRESS VECTORS (BACK).

SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).

Enter plot number and reducing factor in free format.

(2) PRIMARY OR SECONDARY (1 OR 2).

Enter 1 or 2 in free format.

(3) TENSILE OR COMPRESSIVE (1 OR 2).

Enter 1 or 2 in free format.

(4) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

E.13 PGFOR1: Plots radial, tangential and normal force vectors for either static or dynamic analysis.

Terminal Response:

(1) SUBPROGRAM PGFOR1 PROPELLER GRAPHICS PROGRAM FOR PLOTTING FORCE VECTORS (FOR EITHER STATIC OR DYNAMIC ANALYSIS).

(2) (For blade modelled using shell elements i.e. IELEM=0)

PROGRAM FOR PLOTTING FORCE VECTORS IN BLADE COORDINATES

SPECIFY PLOT NUMBER AND PLOT REDUCING FACTOR (PERCENT).

PLOT NUMBER	DEFINITION
1	RADIAL NODAL FORCES.
2	TANGENTIAL NODAL FORCES.
3	NORMAL NODAL FORCES.
4	COMBINED PLOT.

Enter plotting option in free format.

(3) (For blade modelled using 20 noded solid elements i.e. IELEM=1)

PROGRAM FOR PLOTTING FORCE VECTORS IN BLADE COORDINATES.

PLOT NUMBER	DEFINITION
1	RADIAL NODAL FORCES.
2	TANGENTIAL NODAL FORCES.
3	NORMAL NODAL FORCES.
4	COMBINED PLOT.

FACE NUMBER	DEFINITION
1	FRONT (PRESSURE) SURFACE
2	BACK (SUCTION) SURFACE

SPECIFY PLOT NUMBER, FACE NUMBER AND PLOT REDUCING FACTOR.

Enter plotting options in free format.

(4)

```
SPECIFY FORCE UNITS TO APPEAR ON PLOT:
(FOR   UNITS:    NIL   LB.   KIPS.  N.    KN.
       ENTER:    0     1     2      3     4)
```

Select number associated with units of interest and enter in free format.

(5) (For dynamic analysis only.)

```
TIME (SEC) = XXX.
TYPE 1 TO PLOT OR 0 TO SKIP.
```

Enter 1 or 0 in free format. If 1 is entered, loading at time XXX seconds is plotted. If 0 is entered, plotting at this time step is suppressed and response is repeated for the next time step.

(6) (For dynamic analysis only.)

```
NO MORE TIME STEPS BEGIN AGAIN AT TIME STEP #1? (0=NO)
```

This response appears if the user requests another time step after the last remaining time step has been presented. Enter a non-zero number in free format to continue plotting; otherwise enter 0.

(7) Plot appears on screen.

A hard copy may be made at this time. To continue press "Return".

(8) (For dynamic analysis only.)

```
TYPE 1 FOR ANOTHER TIME STEP; OTHERWISE ENTER 0 :
```

Enter 1 or 0 in free format. If 1 is entered, the terminal will respond with one of the dynamic analysis responses. If 0 is entered, the following response appears.

(9)

```
TYPE   1 TO CONTINUE;
       0 TO END THIS PLOTTING OPTION:
```

Enter 1 or 0 in free format. If 0, the program terminates; if 1, plotting options are again displayed.

E.14 PGLOD3: Plots time histories of input pressures, finite element pressures and drag forces.

Terminal Response:

(1)
```
SUBPROGRAM PGLOD3   PROGRAM TO PLOT THE TIME HISTORIES OF THE
                    INPUT AND FINITE ELEMENT PRESSURE AND
                    DRAG FORCES

THERE ARE XX LOADING FUNCTIONS: ENTER DESIRED LOADING
FUNCTION:
```

Enter number representing loading function of interest in free format. This response will not appear if there is only 1 loading function.

(2)
```
PLOT NUMBER               DEFINITION

     1                    BASIC INPUT PRESSURE
     2                    FINITE ELEMENT PRESSURE
     3                    DRAG FORCES

SELECT PLOT NUMBER
```

The first option will display the time history of user input pressure data for the face and back surfaces. The second will display the time history of pressure data interpolated to the finite element grid for the face and back surfaces. The third will display the time history plots of the drag forces applied at the midchord.

(3) (Only for plotting options 1 and 2)

```
SPECIFY PRESSURE UNITS TO APPEAR ON PLOT;
(FOR UNITS:     NIL   PSI   KSI   PSF   KSF   PA    KPA   MPA
ENTER:           0     1     2     3     4     5     6     7)
```

Select number associated with units of interest and enter in free format.

(4) (Only for plotting option 3)

```
SPECIFY FORCE UNITS TO APPEAR ON PLOT;

(FOR   UNITS:   NIL   LB.   KIPS.  N.    KN.
       ENTER:    0     1     2     3     4)
```

Select number associated with units of interest and enter in free format.

(5) RADIAL POSITIONS AVAILABLE:

X.XXXXX X.XXXX X.XXXX

ENTER DESIRED RADIAL POSITION:

If option 1 is chosen, radial positions will be those at which the pressure data was defined by the user. For option 2, radial positions correspond with the finite element grid. For option 3, the radial positions will be those at which the drag forces were applied. This response does not appear for the first two options if pressure loading is uniform.

(6) CHORDWISE POSITIONS AVAILABLE:

X.XXXXX X.XXXXX X.XXXXX

ENTER DESIRED CHORDWISE POSITION.

If option 1 is chosen, chordwise positions will be those at which pressure data was defined by the user. For option 2, chordwise positions correspond with the finite element grid. For the third option, this prompt is skipped as drag forces are assumed to act at midchord.

(7) (Plot appears on screen)

When the border is drawn, the plot is complete. To continue, press return.

(8) ENTER 0 TO END THIS PLOTTING OPTION;
OR 1 TO SPECIFY A NEW PLOT NUMBER;
OR 2 TO SPECIFY A NEW LOADING FUNCTION.

Enter option number in free format. If entry is 0, program PVAST4 continues; if entry is 1, plotting options (2) are again displayed; if entry is 2, a new loading function (1) may be selected. Option #2 will not appear if there is only 1 loading function.

E.15 PGFOR2: Plots time history curves of resultant propeller forces and moments acting on the shaft for a single or multi-blade propeller.

Terminal Response:

(1)
```
PROGRAM PGFOR2     PVAST GRAPHICS PROGRAM FOR PLOTTING TIME
                   HISTORIES OF RESULTANT PROPELLER FORCES
                   AND MOMENTS ACTING ON A PROPELLER SHAFT.
```

(2)
```
SPECIFY FORCE UNITS TO APPEAR ON PLOT;

(FOR   UNITS:  NIL  LB.  KIPS.  N.   KN.
       ENTER:  0    1    2      3    4:)
```

(3)
```
SPECIFY MOMENT UNITS TO APPEAR ON PLOT:

(FOR UNITS:  NIL  LB-IN  LB-FT  KIP-IN  KIP-FT  N-M  N-MM  KN-M  KN-MM
     ENTER:  0    1      2      3       4       5    6     7     8:)
```

Response to (2) and (3) in free format.

(4)
```
THERE ARE XX FORCING FUNCTIONS:
ENTER DESIRED NUMBER
```

This response appears only if a frequency response analysis was performed. Enter number associated with desired frequency in free format.

(5)
```
COMPONENT TO BE PLOTTED:
(FOR COMPONENTS:  FX  FY  FZ  MX  MY  MZ
     ENTER:       1   2   3   4   5   6:)
```

Enter number in free format correspond to desired force or moment to be plotted.

(6)
```
ENTER 0 FOR DEFAULT SCALING OF ORDINATE AXIS:
```

If 0 is entered, the axis will be scaled according to the maximum and minimum values available. If a non-zero number is entered, the user must enter the desired maximum and minimum values.

(7) Plot appears on screen. To continue, press return after plotting is complete.

(8)

```
ENTER  0 TO TERMINATE
       1 TO SPECIFY A NEW COMPONENT TO BE PLOTTED
       2 TO SPECIFY A NEW FORCING FUNCTION
```

If 0 is entered, program execution will end; if (1) is entered, control is transferred to (5); if 2 is entered, control is transferred to (4). Option 2 is not presented if a frequency response analysis was not performed.

APPENDIX F

GENERATION OF A FINITE ELEMENT OR A SURFACE PANEL MODEL FOR THE FLUID SURROUNDING THE BLADE

F.1 Introduction

PVFLU1 generates a single layered thin fluid model using the data on file PREFX.CDR. The thin fluid model option permits the user to check fluid geometry, in particular with respect of the element proportioning, without resorting to hidden line plotting. Provisions also exist for interactive editing of fluid model geometry data, including radius, chord, skew, rake and pitch angle. It is assumed the fluid model will be using the blade element grid to define the interface elements.

PVFLU1 generates a file PREFX.CFM. This file essentially contains the same data as in PREFX.CDR except that it sets the number of layers to 1 and the thickness of the layer to 0.001.

If file PREFX.CDR does not exist or the fluid data is not on the file and if file PREFX.CFM does not exist a message to that effect is issued and the program execution terminates. If run as a PVAST module, control returns to the calling program where the user is given the option to generate the fluid data on PREFX.CDR. The fluid data on the PREFX.CDR file is generated by supplying the data required in Section F.2.

If any changes are made to the fluid data on file PREFX.CDR, PVFLU1 must be rerun to update file PREFX.CFM.

F.2 PVFLU1/PVFLU2/PVFLU3: Input data for generation of fluid model (See Sections 3.2 and 4.3)

Header Card (A6)

FLUID (See Appendix K further details on use of header cards.)

Card 1

IFLUD = Code for specifying type of fluid model. If IFLUD=1, a finite element model is generated. If IFLUD=2, a surface panel model is generated. The original blade finite element grid is used to generate the surface panel model.

Omit CARD 2 if IFLUD = 1.

Card 2 (2I5)

IPTC = Printing code. If IPTC≠0, nodal coordinates and surface panel nodes will be printed.

IBC = Fluid surface condition code. If IBC=0, no free-surface effects are included. If IBC=1 free surface effects in X-Y plane are accounted for.

Omit CARDS 3 to 9 if IFLUD = 2.

Card 3 (3I5)

IGRID = Code for specifying type of grid for fluid/structure interface. If IGRID = 1, original blade elements are used (see Figure F-1). If IGRID = 2 user specified grid (see Figure F-2) is used for the purpose of (i) reducing the problem size (i.e. the size of the added mass matrix) or (ii) generating added mass for master nodes (for substructure analysis). This user specified grid must describe 8-noded interface elements, as shown in Figure 6.16, using nodes already existing on the blade surface. Node selection is accomplished by selecting radial and chordwise positions from those used to define the original blade elements via CARDS 5 and 6. Interface elements are described via CARD 9.

NLAYER = Number of layers of fluid elements on propeller face and back (the total number of layers is 2 x NLAYER).

IPTC = Printing code. If IPTC≠0, nodal coordinates and element nodes will be printed.

Omit CARDS 4, 5, and 6 if IGRID = 1.

Card 4 (2I5)

KP = Number of radial positions to be used to define reduced fluid/structure interface grid (see Figure F-2).

LP = Number of chordwise positions to be used to define reduced fluid/structure interface grid (see Figure F-2).

Card 5 (16I5)

$IRP_{(1)}$ ↓ $IRP_{(KP)}$ = Integer radial array locations, selected from the radial positions used to define the original blade elements.

Card 6 (16I5)

$ICP_{(1)}$ ↓ $ICP_{(LP)}$ = Integer chordwise array locations, selected from the chordwise positions used to define the original blade elements.

Card 7 (2E10.3)

DLEFB = Distance from leading edge to fluid boundary expressed as fraction of full blade radius (see Figure F-3).

DTEFB = Distance from trailing edge to fluid boundary, expressed as fraction of full blade radius (see Figure F-3).

Card 8 (8E10.3)

$DPTHF_{(1)}$ ↓ $DPTHF_{(NLAYER)}$ = Depth of fluid layers 1 through NLAYER, expressed in fraction of the full radius (see Figure F-3).

If IGRID = 2, provide Card 9 for the KP/2 and LP/2 fluid/structure interface elements.

Card 9 (8I4)

N1 = Fluid/structure interface element nodes, defined as shown in Figure 3.9.
↓
N8

Card 10 (E10.3)

DEN = fluid weight per unit volume/g (force-time^2/length^4)

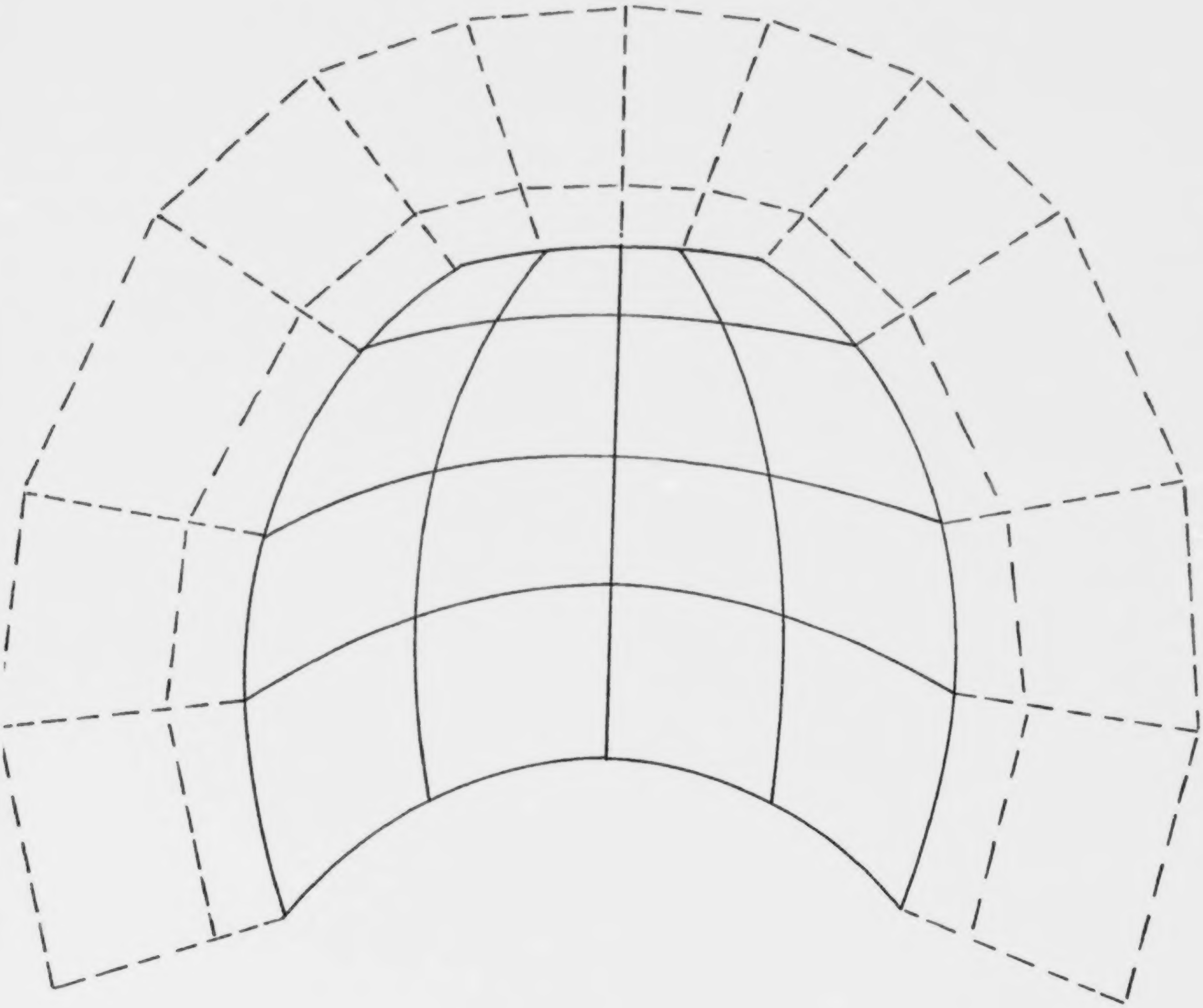
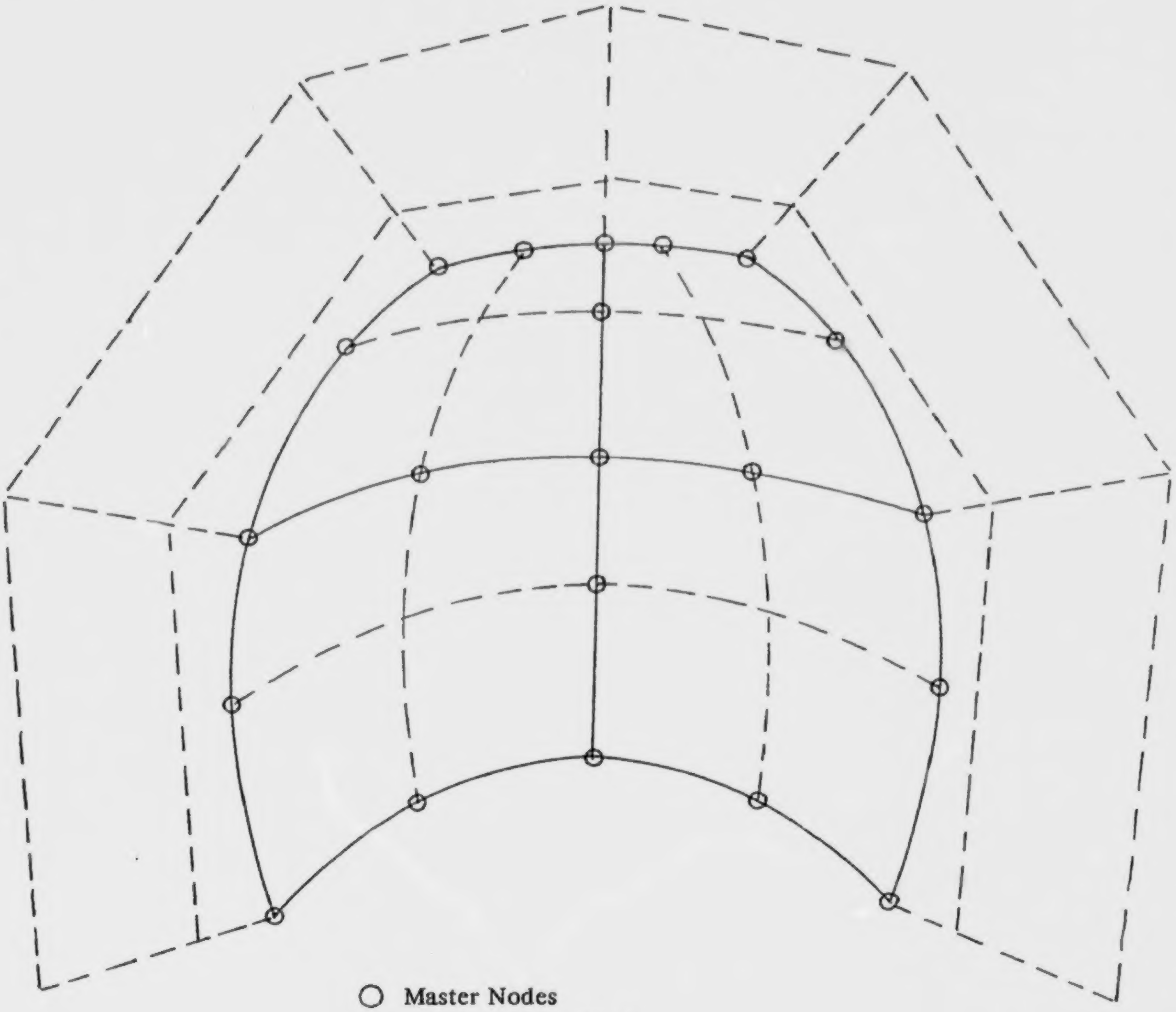


FIGURE F.1: Finite Element Mesh for a Typical Propeller-Fluid Model



○ Master Nodes
----- Original Blade Mesh

FIGURE F.2: Reduced Finite Element Mesh for a Typical Propeller-Fluid Model

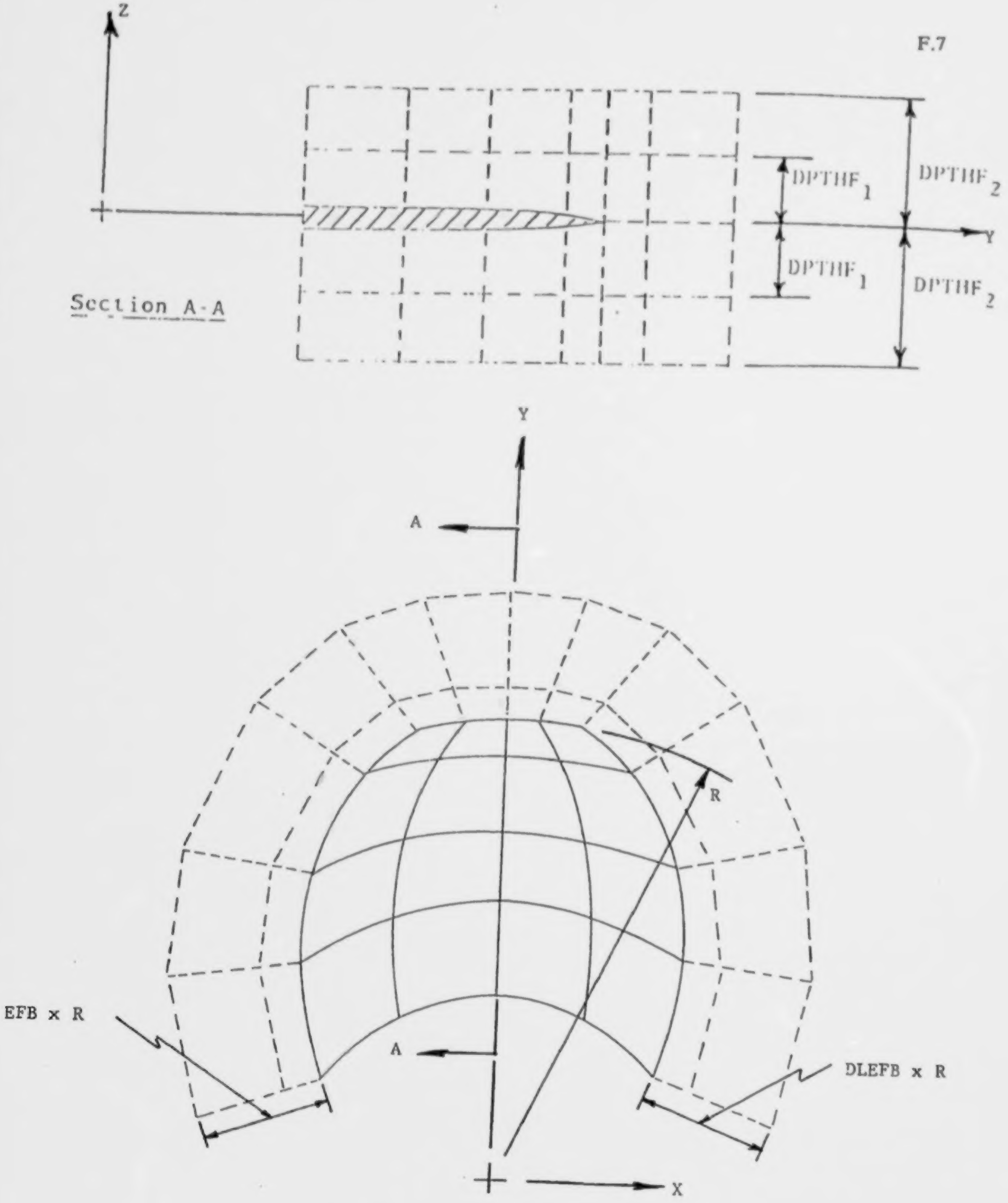

Z
DPTHF$_1$
DPTHF$_2$
Y
DPTHF$_1$
DPTHF$_2$
Section A-A
Y
A
R
EFB x R
A
DLEFB x R
X

FIGURE F.3: Geometry Definitions

APPENDIX G

OPERATING INSTRUCTIONS FOR PROGRAM PGFLUD FOR PLOTTING FLUID MODEL

Terminal Responses:

(1) SUBPROGRAM PGFLUD PLOTS THE FLUID MODEL

SPECIFY PLOT REDUCING FACTOR (PERCENT).

Enter reducing factor in free format. This reducing factor (%) is used to reduce the size of the plot.

(2) SPECIFY ROTATIONS ABOUT Z, Y, AND X AXES.

Enter, in degrees, the desired orientation of the plot. These are real numbers are must be entered in free format. When zeros are specified for all three axes, the global X and Y axes correspond to the screen's horizontal and vertical axes, respectively; the global Z axis is out of the screen.

(3) SPECIFY FIRST AND LAST ELEMENT TO BE PLOTTED.

Enter 2 integers in free format.

(4) Plot appears on screen.

(5) TYPE 1 TO CONTINUE;
0 TO END THIS PLOTTING OPTION:

Enter 1 or 0 in free format.

APPENDIX H

OPERATING INSTRUCTIONS FOR PLOTHD

Hidden Line Removal

Introduction

PLOTHD is a program for displaying 3-dimensional perspective drawings of finite element models with hidden line removal. The program can also display displaced shapes, and displacement and stress contours, also with hidden line removal. PLOTHD also provides an option to randomly section the model and thus to display internal surfaces.

PLOTHD requires as input the files generated by three modules. The first module, POSTV3, takes the displacement and stress files from a VAST execution and processes them for use later on. The second module, VASMOV, takes the model geometry and allows the user to select the parts of the model he wishes to have displayed. The third module is the optional random sectioning program, VASSEC.

As each processing module has a specific function, they will be described in separate sections as follows:

Section 1. describes the displacement/stress processor module (POSTV3)
Section 2. describes the model geometry processor (VASMOV)
Section 3. describes the random sectioning module (VASSEC)
Section 4. describes the hidden line program (PLOTHD)
Section 5. contains a complete example

Section 1: Result Processing - POSTV3

The displacement/stress processor, POSTV3, takes the VAST displacement file PREFX.T52 or the stress files PREFX.P20 and PREFX.P21 and requests the user to select the desired component of displacement or stress and writes the selected data to PREFX.BYU. Only one component of displacement or stress may be selected per run.

The terminal responses for this module are as follows:

```
(1)     SPECIFY FUNCTION FILE TO BE GENERATED:
        ENTER  1 FOR STATIC OR DYNAMIC DISPLACEMENTS
               2 FOR FREQUENCY ANALYSIS DISPLACEMENTS
               3 FOR BUCKLING ANALYSIS DISPLACEMENTS
```

```
4 FOR STATIC OR DYNAMIC STRESSES:
```

If 1, 2, or 3 is entered, response (2) is output; if 4 is entered response (4) is output.

(2)

```
ENTER 5 FOR DISPLACED SHAPE
```

If any number other than 5 is entered, it is assumed that displacement contours are desired and response (3) will therefore be output.

(3)

```
ENTER COMPONENT OF DISPLACEMENT
(1-X, 2-Y, 3-Z, 4-TOTAL)
```

(4)

```
SPECIFY WHICH STRESS COMPONENT YOU WANT:
(FOR STRESS:  SXX  SYY  SZZ  SXY  SYZ  SXZ  SP1  SP2  SP3  SVMH
ENTER:         1    2    3    4    5    6    7    8    9   10):
```

Enter the number corresponding to the desired component.

Section 2: Model Geometry Processing - VASMOV

The model geometry processor, VASMOV, takes the geometry from the VAST files PREFX.USE and/or PREFX.GOM and processes it into a form compatible with the random sectioning module or the plotting module, depending on whether or not random sectioning was requested.

The operation of the module is explained by an example of a typical run. Each response is numbered and explained in the subsequent text. The user input is underlined in the example.

(1)

```
SAMPLE PROPELLER ANALYSIS
```

(2)

```
* 1 SUBSTRUCTURES *
```

(3)

```
SUBSTRUCTURE #1 -  226 GEOMETRIC NODES
                   113 DISPLACEMENT NODES
                   1 ELEMENT GROUPS
```

(4)

```
TYPE   NUM   FIRST   LAST
  1     30     1      30
```

(5)

```
DO YOU WANT TO PLOT ANY ELEMENTS?  (Y/N OR S TO STOP)
```

(6) DO YOU WANT TO PLOT ALL ELEMENTS?

(7) ENTER "R" FOR A RANGE OF ELEMENTS
"I" FOR INDIVIDUAL ELEMENTS
OR ENTER DESIRED ELEMENT TYPE

(8) ENTER FIRST AND LAST ELEMENTS

(9) MORE? ENTER "R", "I", ELEMENT TYPE, ON <CR>

(10) HOW MANY INTERIOR NODES PER EDGE (1 2 3)? 1

Explanation of Previous Example Run

1. The title of the file is printed.

2. The file contains one substructure.

3. Each substructure is processed in turn. The composition of the first is given.

4. Table of limits and size of each element group. Each row of numbers describe, respectively, the element type, the number of the first element in the group, the number of the last element, and the total number of elements in the group. This table can be used by the operator to assist in extracting subsets of the substructure.

5. The substructure may be accepted or rejected by entering Y or N. If the response is N, the next substructure, if any, is processed. If the response is Y, the processing of the current substructure continues.

6. The user is provided with the opportunity to display all elements by response (Y).

7. There are three ways of specifying elements:

   (a) Range of element (contiguous element number (blocks))
   (b) Individual elements; and
   (c) All elements of a particular type.

8. ELEMENT TYPE XX IS CURRENTLY INVALID FOR THIS GRAPHICS PROGRAM. SELECT ONLY THE VALID ELEMENT TYPES. VALID ELEMENT TYPES ARE:

DESCRIPTION	TYPE
THICK-THIN SHELL	1
20-NODE BRICK	2
TRIANGULAR PLATE	4
4-NODE SHELL	5
TRANSITION	6
TRIANGULAR MEMBRANE	9
WARPED-STIFFENED MEMBRANE	11
8-NODE BRICK	16
TETRAHEDRON	17
12-NODE QUADRILATERAL MEMBRANE	18
8-NODE QUADRILATERAL MEMBRANE	20
FRACTURE ELEMENT	21
4-NODE SHEAR WEB	23

This error message is displayed if the model to be plotted contains these element types and they are specified to be displayed.

9. Extract 10 elements starting at number 4.

If the response above had been I, the prompt and a typical response is illustrated in the next two lines.

Enter block numbers (max. of 20)
1 2 3 7 8 9 13 14 15

The numbers need not be consecutive but they must be monotonically increasing.

For either contiguous or individual elements, the extracted set must be of a single type except that elements of type 1 and 6 may be mixed. The monotonic rule, however, always applies. The table illustrated at (4) in the example shows the limits of element types and the number of elements of each type and can be used to assist the user in extracting subsets.

If all the elements of one type are required, an integer indicating the element type is all that is necessary when prompt (8) is given.

10. A fine grid is produced by dividing each 8-node polygon into 4 component quadrilaterals and each 6-node triangle into 4 component triangles. This is useful in displaying models with highly warped faces but the resulting plot will obviously not represent the actual finite element model. Also, as this feature greatly increases the number of elements, the node limit or element limit may be exceeded in this module or subsequent ones. Cautious use of this feature is therefore advised.

11. Each normal (4-sided) face of a 20-node solid element has 8 nodes (4 corner nodes and 4 mid-side nodes). For edges that are straight or nearly straight this is sufficient for a fairly good representation of the face. For curved edges, however, extra nodes along the edge will give a smoother curve. Two internal nodes per side will divide the edge into three equal segments, while three nodes will divide it into four segments.

Section 3: Model Sectioning - VASSEC

If the user has selected random sectioning, the VASSEC module is called. This module allows the user to effectively cut the model any number of times so as to view internal surfaces. To do so, the user specifies a point on or within the model which will lie on the cutting plane and the direction cosines of a vector which is normal to the cutting plane.

As in the previous section, the operation of this module will be explained by a example. Each response is numbered and explained in subsequent text. The user input is underlined.

```
(1)     <PLOTHD SYSTEM PROCESSOR>
        <FOR CLIPPING AND CAPPING OF SOLIDS>

(2)     <READ     1 PARTS    268 NODES    30 ELEMENTS>

(3)     <CLIPPING PASS     1>

(4)     >>

(5)     PLAN
        <ENTER PLANE DEFINITION FOR PARTS 11 THROUGH 12 AS:
        11,12,POINT(X,Y,Z),NORMAL(X,Y,Z)>
        >>
        1 1 0.0 8.0 0.0 1 0 0
        >>
```

```
(6)    PART ID    OLD ELEMENT          LIMITS
          1            1                 30
       PART ID    NEW ELEMENT          LIMITS
          1            1                 18
          2           19                 36

(7)    <CLIPPING PASS     2>
       >> <CR>

(8)    ENTER A TITLE FOR THIS DATA SET
       SECTIONED SAMPLE PROPELLER BLADE

(9)    <GEOM FILE:   2 PARTS    28 NODES    108 ELEMENTS>

(10)   <NUMBER OF DISPLACEMENT FILES>
       >> <CR>

(11)   <NUMBER OF SCALAR FUNCTION FILES>
       >>1
```

Explanation of Example

1. Module identification.

2. A summary of the number of parts, nodes and elements in the model. In finite element terms, a part is a single model or a superelement. In this example, there are no superelements, therefore there is only 1 part. Sectioning will break the model into more than 1 part.

   As the model had been previously processed by the geometry process or (Section 2) the number of nodes may be different than the original VAST geometry.

3. This denotes that the first cut may be made.

4. A blind prompt. Valid input is a carriage return if no cut is required, CENTROID or PLANE. If the model contains elements with a high degree of curvature or warpage, it may be advisable to have the module calculate the coordinates of the centroid of each element face by entering CENT. This will enable the plotting module to more accurately plot the shapes of the element. As this feature increases the number of nodes, care must be taken not to exceed the capacity of the modules. If this feature is desired, it must be entered first.

   Entering PLANE will begin the cutting process.

5. The response if PLANE is entered. Required input is the first and last parts to be cut, the point on the cutting plane, and the direction cosines of the normal to the cutting plane. Since there is initially only one part, parts I1 and I2 are both 1. If extra cuts are made, I1 and I2 will be different. Press return of the next blind prompt. The cutting will be performed.

6. A summary of the old and new model makeup. The original model contained 1 part and 30 elements. The cut model contains 2 parts and 36 elements.

7. The user may enter PLANE again and cut the model in a different manner. In this example, no further cutting was desired so a carriage return was entered.
8. A unique title for the sectional model is requested.

9. A summary of the number of parts, nodes and elements in the sectional model. The numbers of elements in this case does not refer to the solid finite elements but rather to the polygons derived from each face of each solid element. The reason for this is that the plotting module only plots polygons and therefore the form of the geometry must be altered in this manner. Also, as the plotting module only plots the outside surface of each part, all internal polygons and nodes have been discarded.

10. If the displacement/stress module (Section 1) has been run and the displaced shape was requested, enter 1. Otherwise enter a carriage return.

11. If the displacement/stress module (Section 1) has been run and displacement or stress contours was requested, enter 1. Otherwise enter a carriage return.

Section 4: Hidden Line Model Display Program - PLOTHD

After the display file has been created containing model geometry and optionally results, the program enters the plotting module (PLOTHD.BYU). The PLOTHD.BYU operating manual gives detailed instructions for running the module. What follows is a simplified summary of the most important commands available to the user.

PLOTHD.BYU starts with the following display:

```
<PLOTHD SYSTEM DISPLAY>                                                      (1)
<READ: 3 PARTS;  794 COORDINATES;  229 ELEMENTS.>                            (2)
<PREVIOUS RANGE:>                                                            (3)
< - 43.355 <X<  14.074  -0.820  <Y< 49.147  -22.542 <Z< 8.694>               (3)
<ORIGIN MOVED TO: -14.641   24.163   -6.94>                                  (4)
<DISTANCE TO ORIGIN: 201.00, ANGLE: 28.00, ZMIN: 0.10, ZMAXX 402.0>          (5)
< 3 PARTS WITH ELEMENT LIMITS:>
  1  14  15  76   77 229                                                     (6)
>>                                                                           (7)
```

The following points should be noted:

1. PLOTHD.BYU is in control.

2. The display file (PREFX.MOV) has been read and its composition is described. The elements here refer to the polygons in the exterior surface.

3. The original ranges of the XYZ coordinates are listed.

4. The origin of the model has been moved to centre it in the viewing window.

5. A convenient viewing distance, angle of view, and near and far clipping planes have been determined.

6. The model consists of three parts:
   Part 1 consists of elements 1 to 14
   Part 2 consists of elements 15 to 76
   Part 3 consists of elements 77 to 229

7. The symbol >> is a blind prompt. Whenever this occurs, any one of 25 commands may be issued. One of these is HELP which displays a list of all the permissible commands. For example:

```
>>HELP

CENTER    EXIT      HELP    POLYGON  SHIFT      CONTOURS
DEFINE    EXPLODE   IMMUNE  RESTORE  SHRINK     DISPLACEMENTS
DISTANCE  FAST      NODE    RETURN   SUMMARY
DOTTED    FEATURE   PARTS   ROTATE   TRANSLATE
DRAW      FIELD     PIVOT   SCOPE    VIEW

Enter command for further explanation (<CR> to exit).
```

NOTE: Some of the commands are switches, that is when called the first time, the function is enabled. When called a second time it is disabled. These are identified in the explanation.

```
>>>
```

The symbol >>> is a special blind prompt issued by several of the basic commands. It is used when a varying number of sub-commands or parameters are required. The HELP command uses this prompt for explaining the action of the various commands. Simply enter the command to be explained and HELP will print out a short description of its purpose. The purpose of each command (as explained by HELP) will be illustrated in turn. To get out of the sub-command level and return to the basic command prompt>>, a <CR> is entered. Note that the command names are of varying length. When entering a command, the full name may be used, but the first four letters are sufficient in all cases. The abbreviation is used in a few of following examples.

>>>CENTER Centers the model in the display window. This is done automatically when the display file is read in originally but it may be necessary to re-center after certain manipulations of the model (see RESTORE).

>>>DIST Displays the current distance from the observer to the model and allows the user to specify a new distance.

```
>>DISTANCE

<TO ORIGIN (201.00)> 150
```

Whenever parameters are to be changed, the prompt that asks for the new values and displays the current value in parentheses.

Decreasing the distance produces a "zoom" effect.

>>>DOTTED This command is a switch. It allows the user the opportunity to specify dotted lines. Examples:

1. The sequence of commands

```
>>DOTTED
>>DRAW
```

displays the entire model with dotted line.

2. The sequence

```
>>DOTTED
>>FAST
>>DRAW
```

Draws the front faces solid and the back faces dotted.

>>>DRAW Draws a wire model picture (all lines displayed). See also DOTTED, FAST, and VIEW.

>>>EXIT Terminates the session. Compare with RETURN.

>>>EXPLODE Allows the specification of local motion (explosion) of parts.

Example: The model consists of 5 parts stacked one on top of the other (Y direction), with part 1 at the top. Parts 1 and 2 are to be raised a slight amount as a unit, part 3 is to be moved to the left, and parts 4 and 5 are to be lowered as a unit.

```
>>EXPLODE

<PARTS I1/I2, LOCAL MOTION (X,Y,Z)>
>>>   1 2 0 1.5 0
>>>   3 3 -1.5 0 0
>>>   4 5 0 - 1.5 0
>>>   <CR>
```

The local motion distances are based on the same scale as the model. Note the termination of local motion specifications with a <CR>.

```
<LOCAL MOTION SCALE FACTOR (0.000E+00)>1.0
```

Enter the scale factor by which the local motions are to be multiplied. This allows the amount of "explosion" to be controlled without having to re-enter a completely new list of local motions.

After displaying the model, it was found that the separation was too great. So we continue with

```
>>EXPLODE

<PARTS I1/I2, LOCAL MOTION (X,Y,Z)>
>>> <CR>
```

No new local motions are required, enter <CR>.

```
<LOCAL MOTION SCALE FACTOR ( 1.000E+00 )> 0.8
```

Reduce the scale factor by a slight amount.

>>>FAST Invokes the "Poor man's" hidden surface procedure. If called after a DOTTED command, front faces are drawn solid and back faces are dotted.

>>>FEATURE Allows the specification of a "feature" angle. If the dihedral angle of a pair of faces is less than the feature angle, their shared edge will not be drawn. Example:

```
>>FEATURE

<FEATURE ANGLE ( 0.000E+00 )> 10
```

>>>FIELD Allows the user to re-define the frustum of vision.

```
>>FIELD

<ANGLE, ZMIN, ZMAX (29.00 0.10 200.45 )> 40 0.1 200
```

Enter the angle of view, the distance to the near clipping plane (ZMIN), and the distance to the far clipping plane (ZMAX). A small angle of view will reduce the perspective and increase the size of the picture, while a large angle of view will increase the perspective and reduce the size.

>>>HELP Types the available commands on the user's terminal.

>>>IMMUNE Allows the user to make parts temporarily immune to global rotations. This command is designed especially for titles. The immunity only applies to ROTATION commands given after immunity is established and before it is revoked.

```
>>IMMUNE

<PARTS I1/I2 IMMUNE TO ROTATION>
>>> 5 5
>>> <CR>
```

Part 5 is not to be rotated. Note that parts are always specified as a range of consecutive parts. If only one part is to be specified, the number is repeated to indicate a range of 1.

>>>NODE Allows the user to display node numbers. This command is a switch. When given the first time, node numbering is enabled (<NODE NUMBERING ENABLED>). When given again, Node numbering is disabled (<NODE NUMBERING DISABLED>), etc.

NOTE: If a given node appears at more than one location (due to the use of such commands as EXPLODE and SHRINK), the node will be numbered at only one location (i.e. the location with the lowest Y value).

>>>PARTS Allows the selection of all the model or a subset of the model for display.

Example: The model consists of 10 parts. Only parts 4 to 6 are to be displayed.

```
>>PARTS

<PARTS I1/I2 WITHHELD FROM DISPLAY>
>>> 1 3
>>> 7 10
>>> <CR>
```

Parts 1 to 3 and 7 to 10 are to be withheld from the display.

>>>PIVOT

Allows the user to rotate individual parts about an origin defined for that part in the original axis directions of the model. The coordinate triad in the lower left of the picture is designed to help the user remember the original axis orientation. After the IMMUNE command has been used on a part and a subsequent ROTATE command has been given, however, the global coordinate triad no longer coincides with the directions for that part, thus the original model axis orientation must be remembered. Previously defined PIVOT commands remain in effect. A RESTORE command is used to cancel all PIVOT commands.

```
>>PIVOT

<PARTS I1/I2, AXIS,ANGLE>
>>> 1 1 Y 20

>>> 2 2 Y -20
>>> <CR>
```

Enter the part numbers, the axis (X, Y, or Z), and the angle of pivot (i.e. the local rotation).

```
<PARTS I1/I2, RELATIVE ORIGIN>
>>> 1 1 5 0 0
>>> 2 2 -5 0 0
>>> <CR>
```

Enter the part numbers and the relative origin of the part. This specifies the point about which the pivot will take place. Pivots are about axes parallel to the original axes of the model.

>>>POLYGON

This command is a switch (see NODE). When enabled, polygon (face) numbers are displayed.

>>>RESTORE

Zeros rotations, pivots, and translations and initializes rotation and pivot transformation matrices. RESTORE does not modify any parameters set by the EXPLODE command.

NOTE: A CENTER command may be necessary after a RESTORE.

>>>RETURN Returns to the driver module and allows more models to be constructed. Compare with EXIT.

>>>ROTATE Allows the user to rotate the entire model (all parts as a unit) about the translated origin. A right handed coordinate system is used (X to the right, Y up, and Z toward the observer).

```
>>ROTA

<AXIS, ANGLE> Y -60 X 20
```

Enter axis and rotation angle pairs. One or more pairs can be entered on the same line. Previously defined rotations remain in effect, i.e. rotation is relative to a previous ROTATE.

>>>SCOPE Requests information about the picture device.

<DEVICE>

Enter one of the following allowable display device abbreviations:
HPLT (HP PLOTTER)
CPLT (CALCOMP PLOTTER)
TEKT (TEKTRONIX) Default device

<BAUD RATE>

This request is issued for TEKTRONIX display only. This information is required by the PLOT-10 package in order to produce proper delays during screen erasures and hard copy generation. The default rate is 2400.

<COMPUTATION RESOLUTION>

The maximum resolution is 1024 (default value). Reducing the resolution speeds up the drawing of the picture.

```
<SUPPRESS COORDINATE TRIAD?>
```

Enter Y or YES to have the global coordinate triad in the lower left of the picture suppressed. Any other response will cause it to be drawn.

NOTE: The coordinate triad will respond to all ROTATE commands while some (or all) of the parts may be immune to ROTATE (see IMMUNE). The coordinate triad does not respond to PIVOT commands.

There are several other requests for information issued by the SCOPE command. These pertain to continuous tone or colour terminals.

>>>SHIFT Allows the user to move the viewing window in the plane of the display device.

```
>>SHIFT

<SHIFT WINDOW X,Y (0.0  0.0)> 1.0 -0.5
```

Enter the X (to the right) and Y (up) distances by which the window is to be moved. These distances are on the same scale as the model and are absolute, i.e., they are not relative to a previous SHIFT.

>>>SHRINK Shrinks each element (face) of the model towards its centroid, thus separating the faces.

```
>>SHRINK

<CHANGE SHRINK FACTOR FROM (0.000E+00)> 0.2
```

Enter a value greater than or equal to 0.0 and less than 1.0. A zero shrink factor means no shrinkage, a good value is from 0.2 to 0.4.

>>>SUMMARY

Calculates and displays several important ranges and parameters. The minimum and maximum coordinate values are given along with the current location of the origin, distance to the origin, angle of view, location of near and far clipping planes (ZMIN, ZMAX), and the number of parts in the model and their element limits. See also commands DISTANCE and FIELD. This summary is displayed automatically when the display module is first entered.

NOTE: The coordinate ranges do not include the modifications due to rotations and pivots. They are the ranges of the model about its current origin as if no rotations or pivots had been given.

>>>TRAN

Allows the user to shift the origin of the model to a new location.

```
>>TRAN

<ORIGIN FROM ( -14.6 24.2 -6.9 ) TO> -10 24 -7
```

Enter the coordinates of the new origin relative to the original coordinate system of the model. Remember, the view is always down the positive Z axis (in the viewing coordinate system) toward the translated origin.

>>>VIEW

The VIEW command sends the picture defined by all previous commands to the display device. This command will invoke Watkin's algorithm to remove hidden lines and surfaces (see also DRAW command).

>>>CONTOURS

Allows the user to plot displacement or stress contours.

```
>>CONT

<NUMBER OF CONTOURS, LABEL SPACING> 5 20
```

Enter the number of contour lines (26 max) and the label spacing (the number of raster-lines between labels). Labels can be avoided by entering a zero.

```
<RANGE (X1,X2)> 0 16000
```

Enter the maximum and minimum scalar function values (displacement or stress) to be plotted.

>>>DISPLACEMENT Allows the user to plot the displaced shape of the model.

>>DISP

<DISPLACEMENT SCALE FACTOR (DELTA)> 10

The factor by which the displacements are multiplied so that the displaced shape may be seen.

<PLOT DISPLACED SHAPE ONLY?>

If carriage return is entered, the displaced shape will be superimposed over the undisplaced shape.

Enter Y for displaced shape only.

>>> Enter <CR> to get out of the sub-command level and return to the basic command prompt.

NOTES:

After a DRAW or VIEW command, the screen is blanked and the picture is displayed. The issuing of the basic command prompt >> is suppressed at this point to allow the user the opportunity of obtaining a hard copy without the prompt showing on it. In order to continue with more commands, a <CR> must be entered before the prompt re-appears.

Remember the distinction between the two blind prompts >> and >>>. If a command is entered following the basic prompt >>, that command will be executed. If the HELP command is entered and then another command after the prompt >>>, the action of the command will be explained. The special prompt also is used by several other basic commands for requesting a varying list of numeric parameters. To terminate the list, a <CR> is entered.

Section 5: Demonstration Problem

The example model is a sample propeller blade. All user input is underlined and references are made to various illustrations.

A displaced shape file and stress file have been generated by separate runs of the displacement/stress module (Section 1). The example begins with the model geometry processor.

The first series of illustrations has no random sectioning. That feature is shown later.

```
SAMPLE PROPELLER ANALYSIS
* 1 SUBSTRUCTURES *

SUBSTRUCTURE #1 -  226   GEOMETRIC NODES
                   113   DISPLACEMENT NODES
                     1   ELEMENT GROUPS

TYPE    NUM     FIRST     LAST
 1       30       1        30
DO YOU WANT ANY PART OF IT(S TO STOP)? Y
DO YOU WANT A SUBSET? <CR>
* PART NUMBER *

DO YOU WANT A FINE GRID? <CR>
HOW MANY INTERIOR NODES PER EDGE (1 2 3) 1

<PLOTHD SYSTEM DISPLAY>
<READ:    1 PARTS;    330 COORDINATES;   82 ELEMENTS.>

NOTE: DISPLACED SHAPE FILE DFFP1.DSP DOES NOT EXIST.
<PREVIOUS RANGE:>
<   -4.067 <X<   3.810   2.965 <Y<   10.890   -2.787 <Z<   2.787>
<-4.96364E+02 <SCALAR FUNCTION< 6.06474E+04>
<ORIGIN MOVED TO:   -0.129   6.928   0.000>
<DISTANCE TO ORIGIN:   27.74 ,ANGLE:  28.00 ,ZMIN:   0.10 ,ZMAX:   55.47>
< 1 PARTS WITH ELEMENT LIMITS:>
  1  82
```

```
>> VIEW
```

(Plot appears on screen - see Fig. H.1)

```
>>ROTA

<AXIS, ANGLE> Y 45

>>VIEW
```

(Plot appears on screen - see Fig. H.2)

```
>>ROTA

<AXIS, ANGLE> Y 95
```

(Total Y rotation is now 140 degrees)

```
>>CONT

<NUMBER OF CONTOURS, LABEL SPACING> 5 0
<RANGE (X1, X2)> 0 16000
```

(5 Contours; no labels; a stress range of 0 to 160000 psi)

```
>>VIEW
```

(Plot appears on screen - see Fig. H.3)

```
>>CONT

<NUMBER OF CONTOURS, LABEL SPACING> <CR>
```

(Contouring option removed)

```
>>DISP

<DISPLACEMENT SCALE FACTOR(DELTA)> 15

<PLOT DISPLACED SHAPE ONLY?> <CR>
```

(Displaced shape multiplied by a factor of 15; displaced shape superimposed on undisplaced shape)

```
>>VIEW
```

(Plot appears on screen - see Fig. H.4)

```
>>EXIT
```

(Plotting ended)

RANDOM SECTIONING

```
<PLOTHD SYSTEM PROCESSOR>
<FOR CLIPPING AND CAPPING OF SOLIDS>

<READ    1 PARTS    268 NODES    30 ELEMENTS>
<CLIPPING PASS  1>

>>PLAN
<ENTER PLANE DEFINITION FOR PARTS I1 THROUGH I2 AS:
I1,I2,POINT(X,Y,Z),NORMAL(X,Y,Z)>
>>
1 1 0.0 8.0 0.0 1 0 0
>> <CR>
```

(Parts 1 through 1 are to be sectioned; cutting plane passes through point 0.0,8.0,0.0; normal to plane has direction cosines 1,0,0)

```
PART  ID  OLD ELEMENT   LIMITS
       1       1          30
PART  ID  NEW ELEMENT   LIMITS
       1       1          18
       2      19          36
```

(Summary of original and updated model)

```
<CLIPPING PASS 2>
>> <CR>
```

(Option to section again; user ends sectioning)

```
ENTER A TITLE FOR THIS DATA SET
```

```
SECTIONED SAMPLE PROPELLER
```

(Title for this session)

```
<GEOM FILE:   2 PARTS    281 NODES    108 ELEMENTS>
```

(Summary of polygon data on output data file)

```
<NUMBER OF DISPLACEMENT FILES>

>> <CR>
<NUMBER OF SCALAR FUNCTION FILES>
```

(No displacement data; 1 file containing stress data)

```
<PLOTHD SYSTEM DISPLAY>
<READ:    2 PARTS    281 COORDINATES:    108 ELEMENTS.>
<PREVIOUS RANGE:>
<    -4.067 <X<   3.810   2.9675 <Y<   10.890   -2.787 <Z<    2.787>
<-4.96360E+02 <SCALAR FUNCTION< 6.06470E+04>
<ORIGIN MOVED TO:   -0.129     6.928     0.000>
<DISTANCE TO ORIGIN:   27.74 ,ANGLE: 28.00 ,ZMIN:   0.10 ,ZMAX: 55.47>
<2 PARTS WITH ELEMENT LIMITS:>
 1  54  55 108

>>ROTA
<AXIS, ANGLE> Y 140
>>CONT
<NUMBER OF CONTOURS, LABEL SPACING> 5 0
<RANGE (X1,X2)> 0 16000
>>VIEW
```

(Plot of sectioned model appears on screen - see Fig. H.5; note difference between Fig. H.5 and Fig. H.3)

```
>>EXPL
<PARTS I1/I2, LOCAL MOTION (X,Y,Z)>
>>  1 1 -2 0 0
>> <CR>
<LOCAL MOTION SCALE FACTOR (0.000E+00)> 1
```

(Part 1 moved 2 units in minus x-direction; movement scaled by a factor of 1)

```
>>VIEW
```

(Plot appears on screen - see Fig. H.6)

```
>>PART
<PARTS I1/I2 WITHHELD FROM DISPLAY>
>> 1 1
>> <CR>
```

(Part 1 withheld from display)

```
>>VIEW
```

(Plot appears on screen - see Fig. H.7)

```
>>EXIT
```

(Plotting ended)

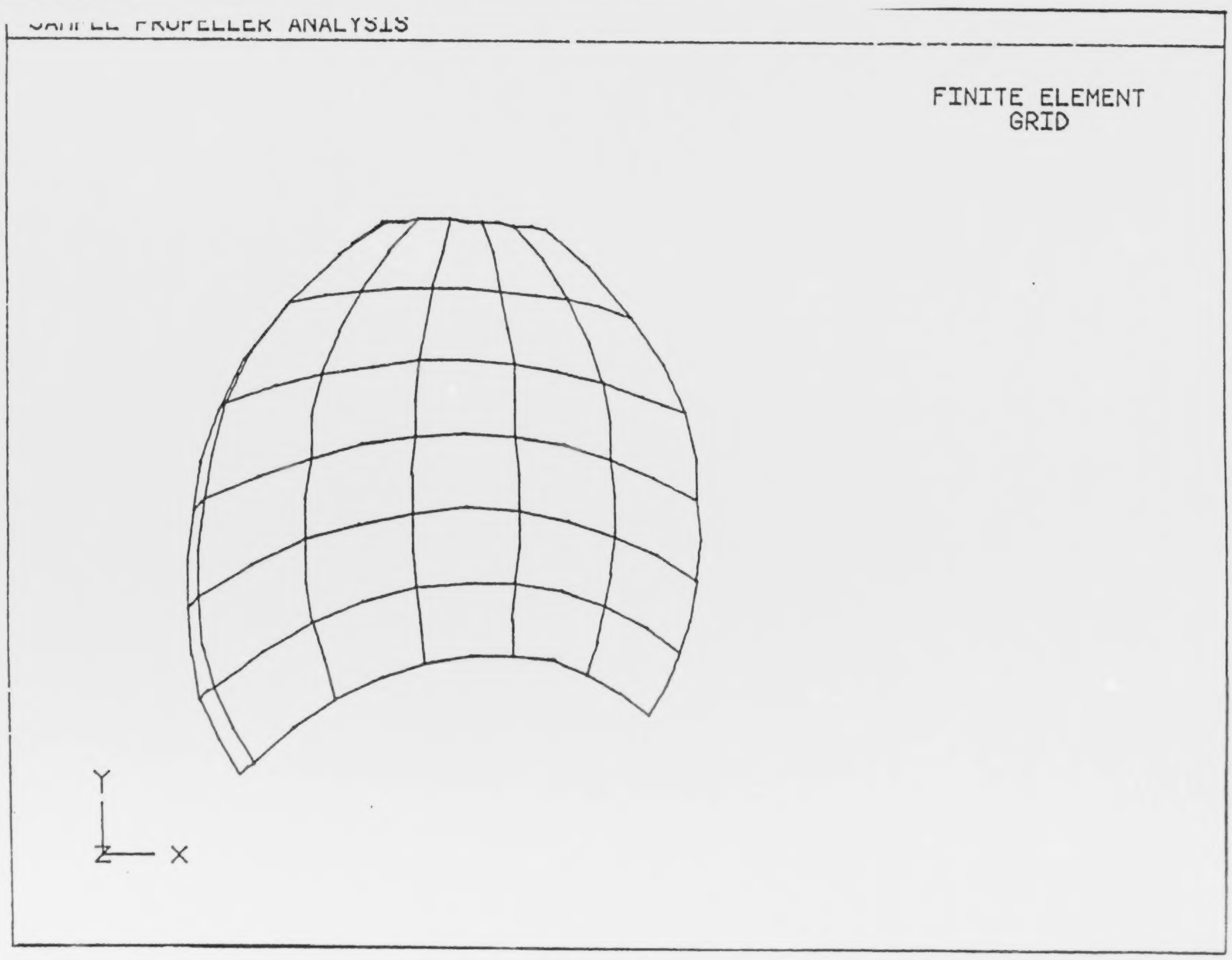

SAMPLE PROPELLER ANALYSIS
FINITE ELEMENT
GRID
Y
Z
X

FIGURE H.1: Basic Model

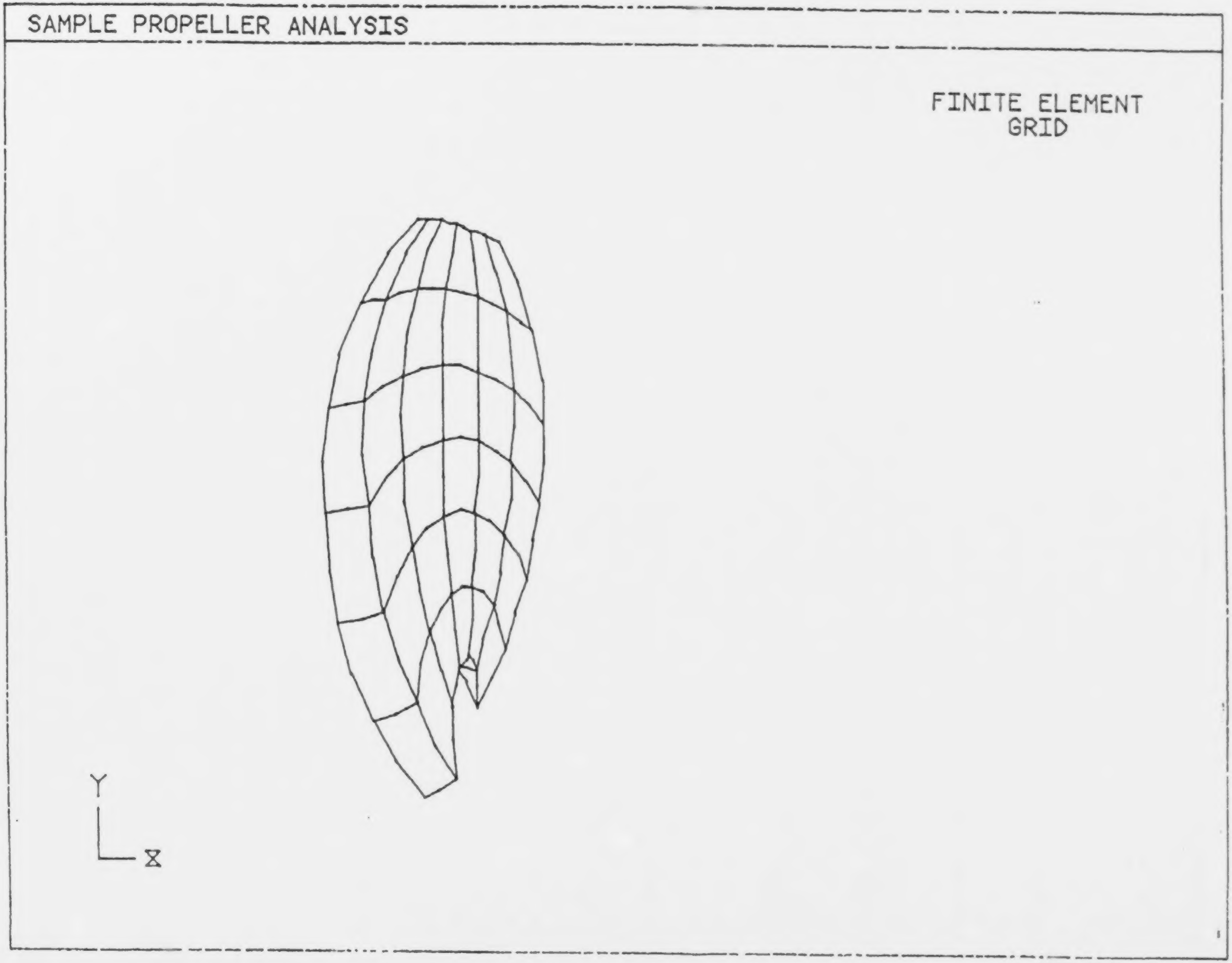

SAMPLE PROPELLER ANALYSIS
FINITE ELEMENT GRID
Y
X

FIGURE H.2: Model Rotated 45° about Y

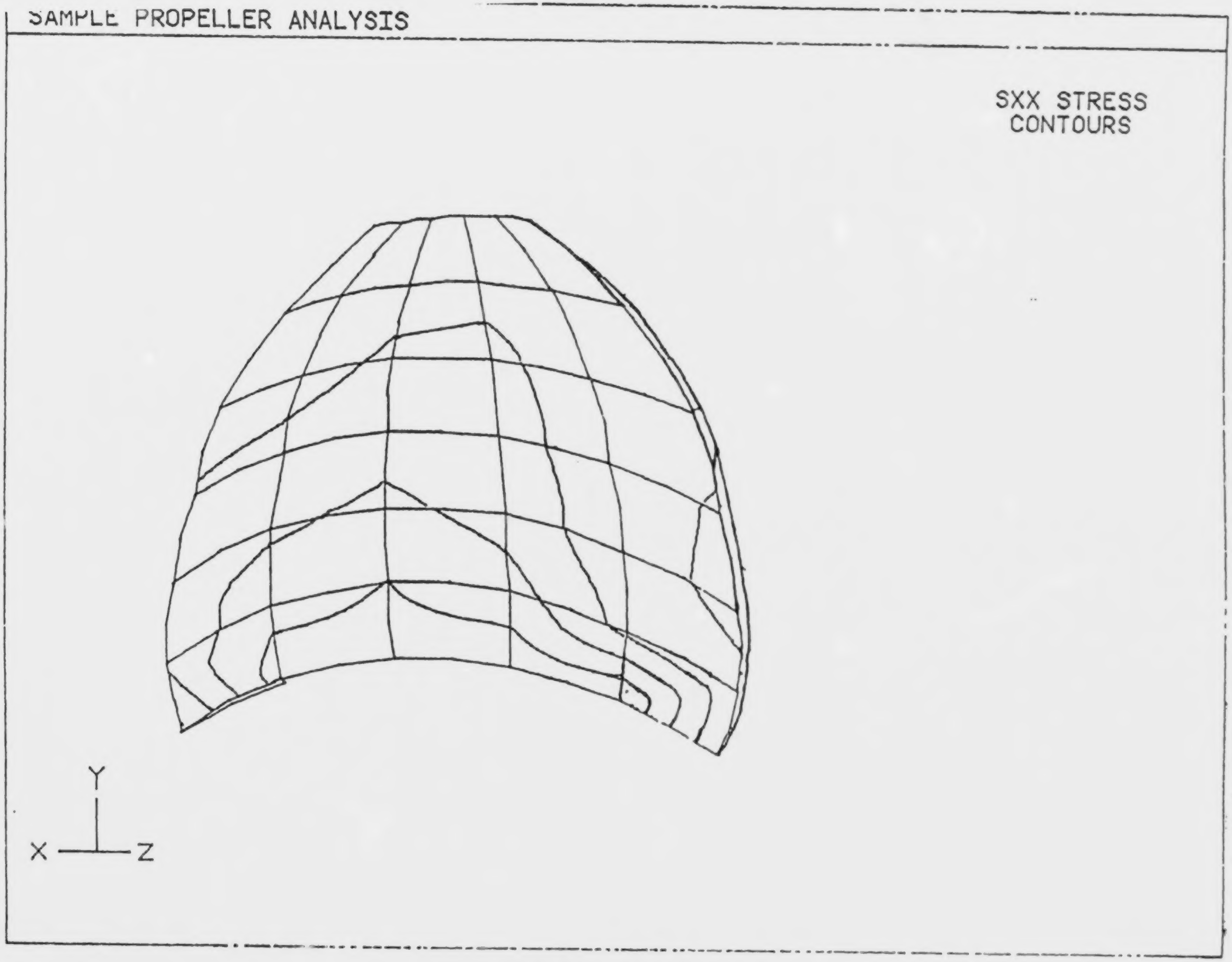

SAMPLE PROPELLER ANALYSIS
SXX STRESS
CONTOURS
Y
X
Z

FIGURE H.3: Stress Contours

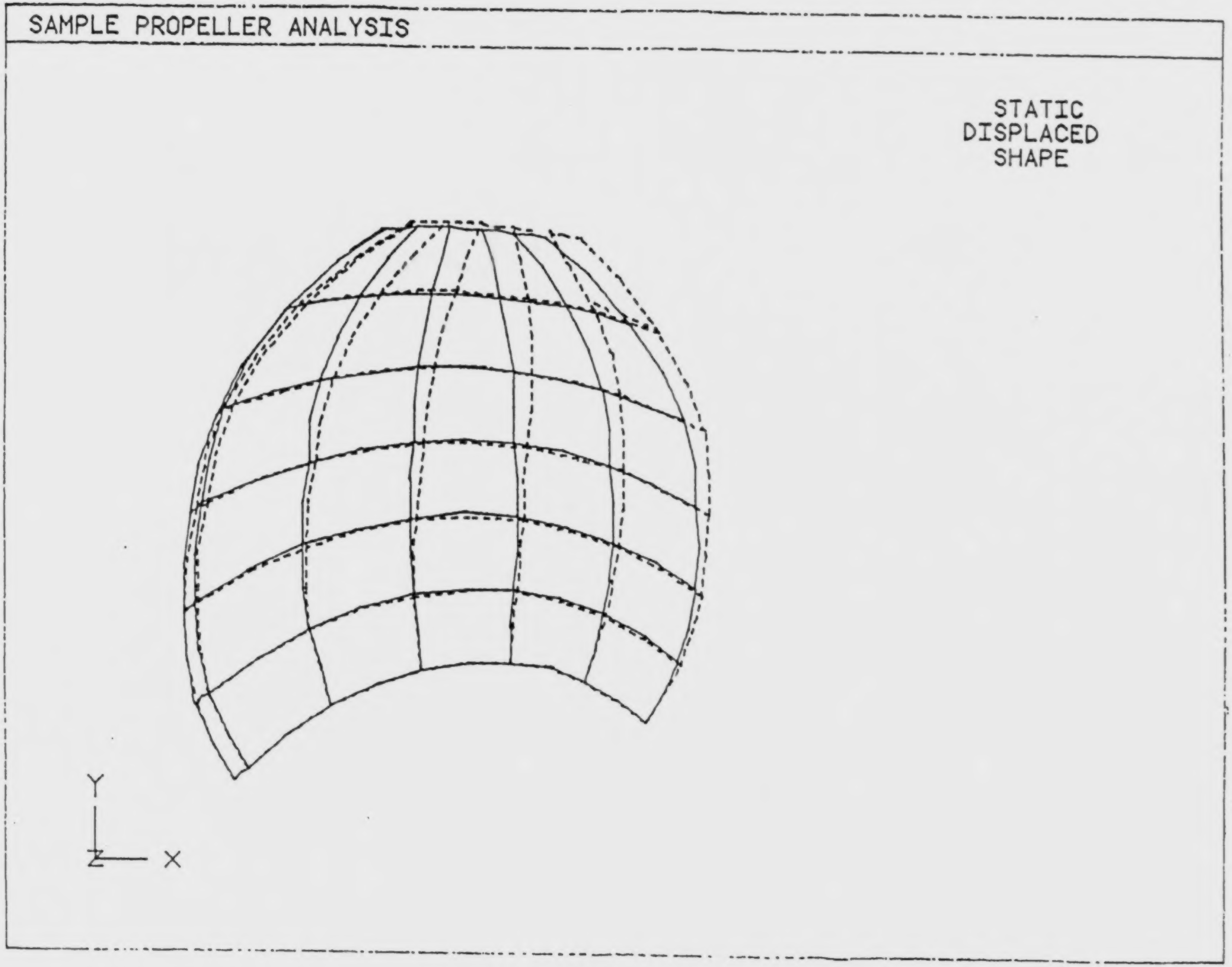

SAMPLE PROPELLER ANALYSIS
STATIC
DISPLACED
SHAPE
Y
Z
X

FIGURE H.4: Displaced Sh

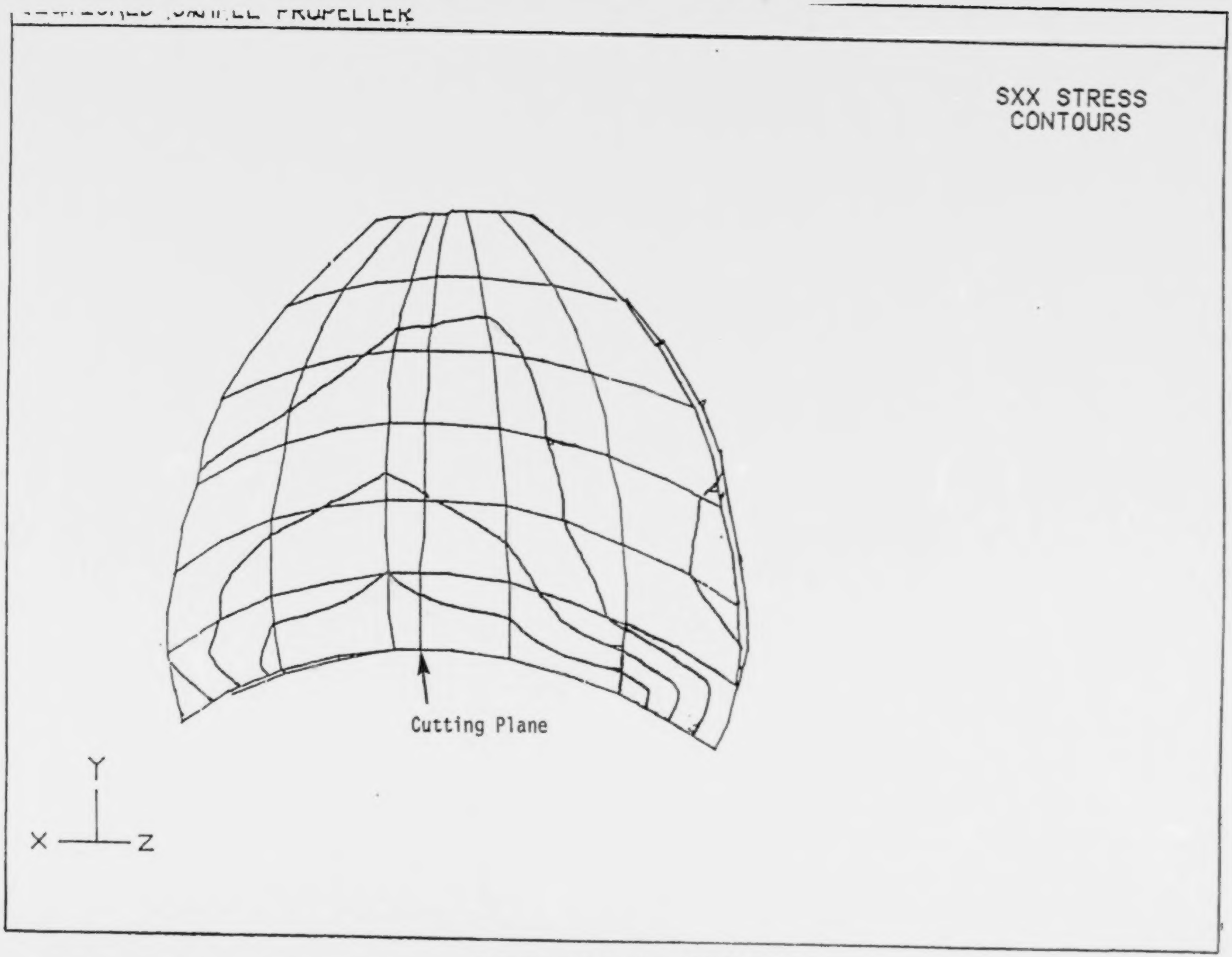

PROPELLER
SXX STRESS
CONTOURS
Cutting Plane
Y
X
Z

FIGURE H.5: Sectioned Model

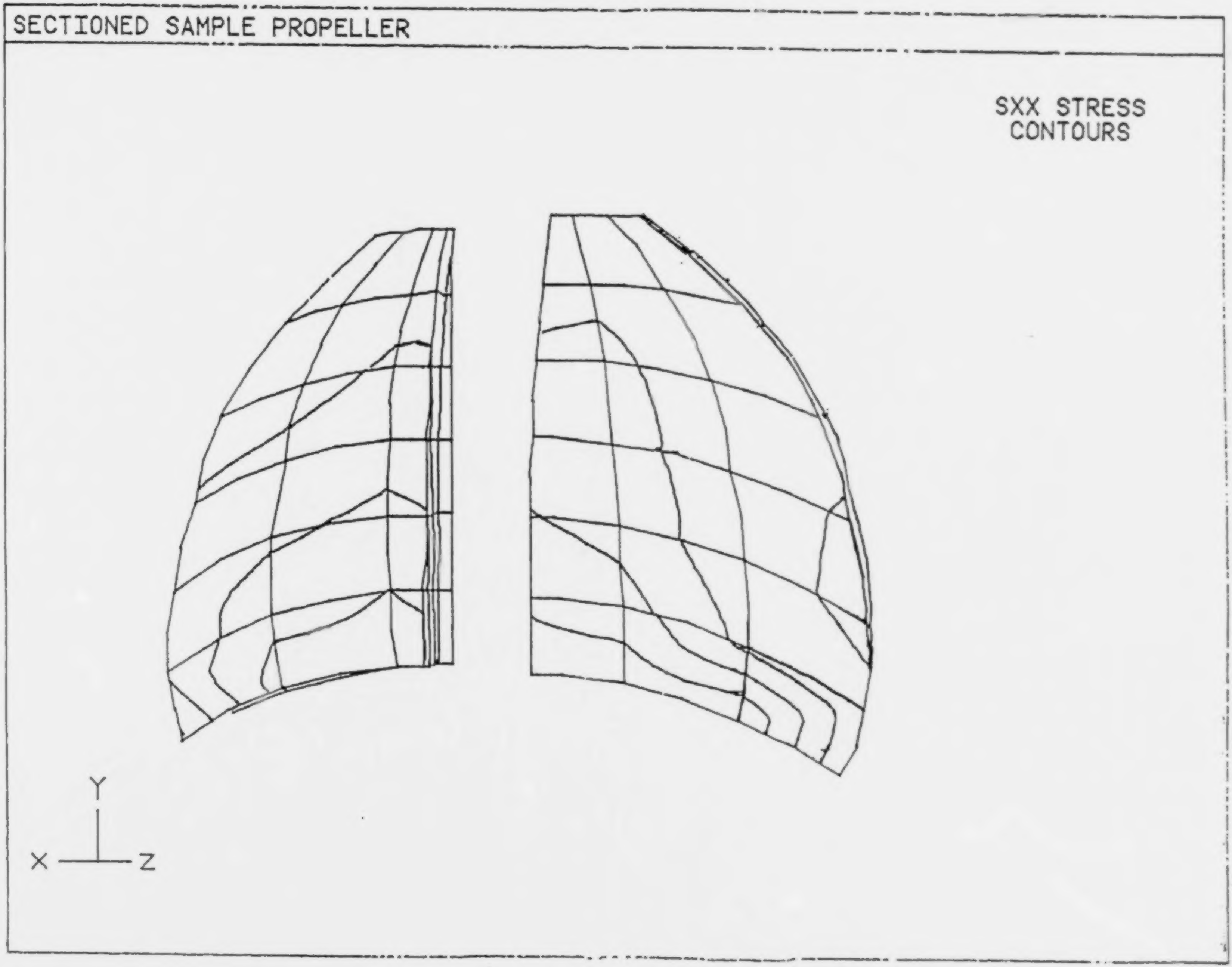

SECTIONED SAMPLE PROPELLER
SXX STRESS CONTOURS
Y
X
Z

FIGURE H.6. Exploded Model

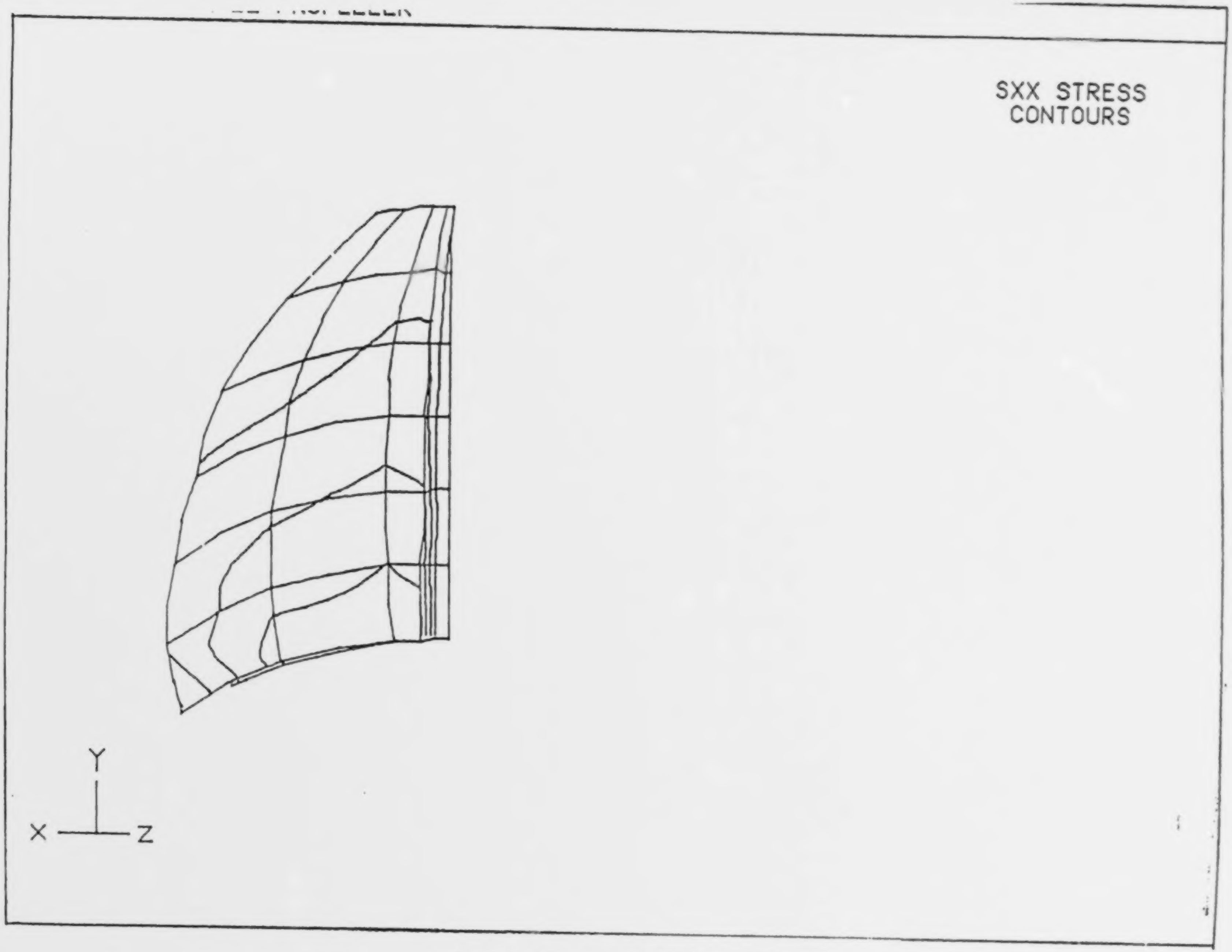

SXX STRESS
CONTOURS
Y
X
Z

FIGURE H.7: Part 1 Withheld from Plot

APPENDIX I

BLADE GEOMETRY GENERATION

This section of PVAST4 consists of three segments and their functions are:

BLADAT - keyboard input or alter blade geometry data.
BLADGM - create complete blade geometry files.
BLADPL - plot blade geometry data.

* Input Data Description *

For further information on these blade geometry generation programs, the reader is referred to independent documentation. [D.R. SMITH and T.R. MacFARLANE. BLADE: A Propeller Blade Geometry Generator Program User's Manual. DREA Technical Communication 91/302, February 1991.]

APPENDIX J

OPERATING INSTRUCTIONS FOR SUBPROGRAM PVUSE

This subprogram generates the .USE and .CTL files required to submit the propeller analysis to VAST.

Static Analysis:

If all data has been generated and no errors are detected a static analysis is transparent to the user except for the statement:

```
#1      FOR VAST ANALYSIS TYPE:

        @PREFX.CTL
        to run VAST interactively, or

        SUBMIT PREFX.CTL
        to run VAST by batch process on the VAX system.
```

where PREFX is the five character alpha-numeric name used to identify files.

This is the correct VAST analysis SUBMIT command for running on the VAX/VMS system. Control now exits this subroutine and returns to the main program.

If superelements are used: Error messages E1 and E2 may appear.

```
E1      ***ERROR*** DIMENSION OF MASTER NODE ARRAY MN1 IN PVUSE IS LESS
        THAN__
        MAX. ALLOWED IS __
        FILE__.USE WILL BE INCOMPLETE FILE __.CTL WILL NOT BE GENERATED.

E2      NO SUPERELEMENT DATA ON FILE __.__
```

If bandwidth reduction is to be used. (Not presently used as programmed.)

```
W1      ENTER THE MAXIMUM NUMBER OF TIMES THE ROWS WILL BE
        REARRANGED FOR BANDWIDTH REDUCTION. (USUALLY 2)
```

W2 ENTER MAXIMUM NUMBER OF ROW-COLUMN INTERCHANGES FOR BANDWIDTH REDUCTION. (USUALLY 1000)

Boundary conditions, normally transparent to the user. If no boundary condition data is to be found:

B1 NO BOUNDARY CONDITION CODE DATA FOUND ON FILE REFERENCE PVAST MANUAL PAGE C6-4 ENTER NUMBER OF DISPLACEMENT NODES TO BE ASSIGNED SPRING-TYPE STIFFNESSES

B2 ENTER DEFAULT VALUE FOR SPRING STIFFNESS. (USUALLY 10E20 IF USING INCHES OR 10E22 IF USING MILLIMETRES)

B3 CODES SPECIFYING SPRING STIFFNESSES FOR 6 DEGREES OF FREEDOM
0 NO SPRING IS ASSIGNED
1 A SPRING IS ASSIGNED

B4 ENTER DISPLACEMENT NODE NUMBER AND 6 CODES (NODE,IDX,IDY,IDZ,MDX,MDY,MDZ)

(B4 is repeated for each node.)

Natural Frequency Analysis:

If superelements are used, error codes E1 and E2 are active.

If bandwidth reduction is used W1 and W2 are presented.

Boundary conditions are checked. Normally transparent

M1 ENTER THE NUMBER OF NODES WHERE LUMPED MASSES ARE LOCATED
ENTER 0 FOR WATER MASS ONLY OR NO. LUMPED MASSES

If 0 is entered, messages M2 to M4 are skipped.

M2 ENTER NODE NUMBER WHICH IS ASSIGNED LUMPED MASSES

M3 AT NODE _____ ENTER TRANSLATIONAL MASSES FOR X,Y,Z (FORCE-TIME**2/LENGTH), (3 NUMBERS)

M4 ENTER ROTATIONAL MASSES ABOUT X,Y,Z AXES (FORCE-LENGTH-TIME**2), (3 NUMBERS)

M2 to M4 are repeated for each node where lumped masses are located.

M5 OUTPUT FILES __.USE AND __.CTL ARE SET TO CARRY OUT ANALYSIS IN WATER ONLY.

FOR IN AIR ANALYSIS, AFTER WATER ANALYSIS IS DONE:

EDIT FILE __.USE TO SET MASTER CONTROL CODES, IELEMS, IBANDR, IASSEM, AND ISTIFM = 0 AND AFTER DATA WORD IMASSM SET IADMS = 0 (PRESENTLY = 1).

EDIT CONTROL FILE __.CTL TO REMOVE FOUR LINES: @RUN ADMASS TO @DELETE __.A* SET THE VAST DATA FILES REQUIRED FOR PLOTTING INTO TEMPORARY FILE NAMES, (EG. FOR SINGLE BLADE NATURAL FREQUENCIES FILE .T51 TO .T51W) FILE .T41 WILL NOT BE ALTERED. AND SUBMIT FOR EXECUTION AGAIN

F1 REFERENCE VAST MANUAL PAGE C9-3 FOR EIGENVALUE ANALYSIS HOW MANY MODES ARE TO BE COMPUTED.

F2 ENTER MAXIMUM NUMBER OF ITERATIONS ALLOWED.(DEFAULT 20)

F3 ENTER ITERATION TOLERANCE. (DEFAULT 0.001)

#1 is now presented.

Dynamic Analysis:

D1 ENTER METHOD OF DYNAMIC ANALYSIS

1 = ANALYSIS BY MODAL SUPERPOSITION ASSIGNMENT OF LAST MODE WILL BE NEEDED. REF C11-6
2 = ANALYSIS BY DIRECT INTEGRATION REFERENCE VAST MANUAL PAGES B-17,C8-1,2,C11-1,11

If superelements are used, error codes E1 and E2 are active.

If bandwidth reduction is used W1 and W2 are presented.

Boundary conditions are checked. Normally transparent to the user. Error coding B1 to B4 may be presented.

M1 mass data is presented.

M2 to M5 may be presented.

If 1 was entered at D1:

F1 to F3 are presented.

D4 REFERENCE VAST MANUAL PAGE C11-6
ENTER NUMBER OF LAST MODE TO BE USED FOR MODAL SUPERPOSITION

D5 ENTER SOLUTION TIME STEP FOR NUMERICAL INTEGRATION (GENERALLY PERIOD OF LAST FREQUENCY/10.)

D6 ENTER NUMBER OF TIME INTERVALS, NH, FOR WHICH DYNAMIC DISPLACEMENTS ARE TO BE COMPUTED. .__* NH MUST BE LESS THAN TOTAL TIME OF APPLIED LOAD.

D7 ENTER DAMPING RATIO FOR MODE (VALUES FROM 0.0 TO 1.0)

(D7 is repeated for each mode selected)

S1 to S3 are now presented.

#1 is now presented

If 2 was entered at D1:

Frequency Response Analysis:

If superelements are used, error codes E1 and E2 are active.

If bandwidth reduction is used W1 and W2 are presented.

Boundary conditions are checked. Normally transparent to the user. Error coding B1 to B4 may be presented.

M1 is presented.

M2 to M5 may be presented.

F1 to F3 are presented.

D9 REFERENCE VAST MANUAL PAGE C11-16
ENTER LAST MODE TO BE USED IN FREQUENCY RESPONSE ANALYSIS

D10 SELECT THE TYPE OF STEADY STATE DISPLACEMENT LOADING:
ENTER 1 FOR SINUSOIDAL LOADING.
2 FOR FOURIER SERIES REPRESENTATION OF PERIODIC LOADING.

D7 is presented.

If 2 was entered in D10, go to D18.

If 1 was entered at D10:

D11 ENTER NUMBER OF LOAD LEVEL FACTORS TO BE USED TO SCALE THE ORIGINAL LOAD VECTORS. (MAX. 50)

Omit if 0 was entered in D11:

D12 TABLE OF _ FREQUENCIES VS LOAD LEVEL FACTORS: ENTER FREQUENCY (CYCLES/SEC.) AND LOAD LEVEL FACTOR

D13__>

Prompt will appear for each set.

D14 ENTER THE NUMBER OF FREQUENCIES IN ADDITION TO THE NATURAL FREQUENCIES FOR DISPLACEMENTS. (MAX. 100)

If D14 was nonzero the following statements D15 to D17 may be presented.

D15 ENTER 1 TO INPUT ADDITIONAL FREQUENCIES
2 TO CALCULATE ADDITIONAL FREQUENCIES USING LOG INCREMENT

If 1 was entered in D15:

D16 ENTER ADDITIONAL FREQUENCIES. (CYCLES/SEC.)

If 2 was entered in D15:

D17 ENTER LOWEST AND HIGHEST FREQUENCIES TO BE USED. (CYCLES/SEC.)

S1 to S4 will be presented.

#1 is now presented

If 2 was entered in D10:

#1 is now presented

Stress Data Responses:

S1 REFERENCE VAST MANUAL PAGE C12-2

S2 ENTER INCREMENT IN TIME INTERVALS AT WHICH DYNAMIC STRESSES ARE TO BE COMPUTED

S3 ENTER 1 FOR TABLES OF MAXIMUM STRESS VALUES TO BE PRINTED
0 FOR NO TABLES

S4 ENTER THE NUMBER OF THE FORCING FREQUENCY AT WHICH THE STRESS HISTORIES ARE COMPUTED

APPENDIX K

PRE-PROCESSING PROGRAM PVAST4

The processing programs shown in Figure 5.1 (with the exception of PVBLD3) have been assembled into one program which permits the user to perform the pre-processing interactively.

The control routine for this program is called PVAST4. When run it allows the user to input and edit subprogram data and generate finite element models, load and boundary data. The data is then written to the appropriate VAST files. PVAST4 also permits the plotting of intermediate results so that the user may adjust his data if necessary.

A restart feature is present in PVAST4 which will, if the user has terminated the program before pre-processing is complete, save all files generated to that point. The user may then restart the program and carry on from the point of termination.

Prior to starting PVAST4, the user must create a file, PREFX.CDR defining the geometry of the finite element model. It thus contains data necessary to run the sub-programs (PVBLD1, PVFIL/HUB/PALM, PVBLD2, PVSUB1/2, PVFLU2/3, PVLODB and PVLODV) which are called. For any propeller model with the exception of the hub keyway model, the basic format for PREFX.CDR generally is as follows:

BLADE1 (A6)
Input data for PVBLD1 (Appendix B.1)
ROOT (A6)
Input data for PVFIL/HUB/PALM (Appendix B.2)
BLADE2 (A6)
Input data for PVBLD2 (Appendix B.3)
SUBMOD (A6)
Input data for PVSUB1/2 (Appendix B.5)
FLUID (A6)
Input data for PVFLU2/3 (Appendix F.2/F.3)
LOADV (A6)
Input data for PVLODV (Appendix C.3)
BOLTS (A6)
Input data for PVBLTA/B/C (Appendix B.8)
ACHANN (A6)
Input data for PVCHL1 (Appendix N)

In addition another independent file PREFX.LIN defining static or dynamic blade pressure loads is to be defined with data formatted as input to subprogram PVLODB (Appendix C.2).

For a two dimensional finite element model for hubs with keyways, the basic format for the file PREFX.CDR is simply

KYHUB (A6)
Input data for PVKEY (Appendix B.6).

Prior to executing each program module required for a particular analysis option, the main program of PVAST4 checks for the existence of relevant data on the file PREFX.CDR (by searching for the associated header). If it does not exist, the user is offered an option of preparing the input data using the interactive program PVEDIT. However, generally a considerable amount of data is required for an analysis and unless sufficient time is available for preparing all data at a single sitting, the procedure of advance preparation of the above indicated data is recommended. PVEDIT may also be used to edit and modify input data but it must be emphasized that certain types of errors are difficult to correct, especially those which result in the number of input records appearing to be incorrect.

To run PVAST4, all sub-programs must be compiled and then linked. The command for compiling a typical sub-program PVSUB.SR4 on the VAX/VMS system is:

```
FOR PVSUB.SR4/N00PT
```

To link PVAST4 on the VAX, the user must first create a link file called typically PVAST4.LNK. The command to initiate the link process would be:

```
@PVAST4.LNK
```

To execute PVAST4, the following command would be used:

```
RUN [DIRECTORY] PVAST4
```

where [DIRECTORY] is the directory in which the file PVAST4.EXE generated by the @PVAST4.LNK command is situated. If the program is being run from the same directory as the PVAST.EXE file or the command "PVAST4" is defined globally, then the user only needs to type PVAST4 to run the program.

APPENDIX L

POST-PROCESSING PROGRAM PPOST4

The processing programs shown in Figure 5.2 have been assembled into one overlayed program which permits the user to perform the post-processing interactively.

The control routine for this program is called PPOSTM. When run it checks for the existence of various VAST output files and then offers an appropriate menu of plotting options available based on the files found. In addition to the output files created by VAST, PPOST4 also requires PREFX.T10 and PREFX.T11 which are created by PVAST4.

To run PPOST4 all sub-programs must be compiled and then loaded into the overlay. The command for compiling a typical sub-program PGSUB.SR4 on the VAX/VMS system is as follows:

```
FOR PGSUB.SR4/N00PT
```

To link PPOST4 on VAX/VMS system, the user must create a link file called typically PPOST4.LNK which instructs the computer as to which programs to be linked. The command to initiate linking of PPOST4 would be:

```
@PPOST4.LNK
```

To execute PPOST4, the following command would be used:

```
RUN [DIRECTORY] PPOST4
```

where [DIRECTORY] is the directory where the file PPOST4.EXE generated by the @PPOST4.LNK command is situated. If the program is being run from the same directory as the PVAST4.EXE file or the command "PPOST4" is defined globally, then the user only needs to type PPOST4 to run the program.

APPENDIX M

THE GEOMETRY OF MARINE PROPELLERS

1. Introduction

A marine propeller is a complex three-dimensional form, the shape of which must be described adequately for strength determination or manufacture. The approach generally taken is to define the geometry of a single blade in detail including the intersection with the hub.

Many different blade shapes have been developed for reasons such as the reduction of unsteady hydrodynamic loading on the blades when the propeller is operating in a highly non-uniform wake behind the ship's hull. Thus, the method chosen for geometry definition must be able to take a wide variety of blade forms into account.

Propeller blade geometry is generally supplied as two-dimensional data in the form of blade sections from which three-dimensional spatial coordinates must be generated. Specification of the blade shape can be complicated by the fact that no one universal format and axis system have been established by blade designers for presenting the geometry data. Therefore, the method developed to generate the spatial coordinates required for structural analysis, manufacture and measurement must be easily adapted to the various forms in which the data is presented.

This manual describes in detail the geometry of propellers and an exact method for converting the two-dimensional data into three-dimensional coordinates. The method can handle blade geometry in the form of NACA section thickness and camberline data or in the form of individual section coordinates. In regions such as the leading edge and the blade tip, where coordinate information is frequently insufficient, the procedure has been further developed to fit acceptable faired curves.

To reduce the effort in producing propeller geometry data, a computerized procedure (BLADGM) has been developed based on the method of geometry definition described. It can be used to generate three-dimensional blade coordinates and to display graphically the input data and the propeller blade shape according to naval architectural convention.

2. Propeller Geometry

2.1 Blade Description

Marine propellers are best described by defining their parts. The four bladed variable pitch propeller shown in Figure M.1 illustrates the major propeller parts. Variable pitch propellers can rotate each blade around its spindle axis; otherwise, they are similar in geometry to fixed pitch propellers except at the blade hub intersection. This intersection portion of a variable pitch propeller is referred to as the blade palm and is a circular base centred on the spindle axis. The palm generally forms a portion of the spherical shaped hub. A sphere is used to ensure a good hydrodynamic surface at the blade hub intersection regardless of the angle at which the blade is set.

On the other hand, fixed pitched propeller blades intersect with a hub that can assume any shape which can generally be defined by a splined cubic curve. In this case, the blades are cast integral with the hub or they are bolted to it.

A propeller blade has two main hydrodynamic surfaces. The surface of the blade which faces aft is referred to as the face of the blade. The surface which faces forward is called the back of the blade. A propeller which normally rotates clockwise when viewed from astern, is called right handed and the leading edge is that edge to the right formed by the intersection of the face and back surfaces. The trailing edge is the edge to the left. The reverse is true for counterclockwise rotating propellers generally referred to as left handed propellers.

The tip of the blade joins the leading edge to the trailing edge at the intersection of the face and back surfaces at the maximum radius from the center of rotation from which the blade extends.

To make a smooth transition from the blade to the hub or palm, a fillet is used. The fillet is most easily formed by a series of single radius curves, the radii becoming a maximum near mid chord and a minimum at the leading and trailing edges. Compound curves have also been used in attempts to produce constant stress fillets. These are described in more detail later in section 2.10.

The assembly of blades forming a propeller has often been referred to as a screw, as in the case of an airscrew for an aircraft, because they are similar in concept. A propeller blade is generated from a series of fully developed sections such as the one shown in Figure M.2. Each of the sections forming the blade is located at a specified radius R_i from the center of rotation. Each section is then rotated about its vertical or spindle axis to a prescribed pitch angle and then wrapped or curved so as to fit on a fictitious cylinder with a radius equivalent to the specified radius for the section. The center line of the section thus forms part of a helix on the cylinder in the same manner as a screw thread. By treating each of the sections in this manner and filling and blending between the sections, two smooth and continuous surfaces are produced forming the face and back of the blade.

2.2 Blade Coordinate Systems

The coordinate system for a propeller blade in most cases is specified in the from of right handed XYZ cartesian coordinates. Unfortunately, there is not a uniform standard for propeller coordinate systems, as shown in Figure M.3. The DREA axis system (Figure M.3a) is used in PVAST. It has the Z axis positive aft coinciding with the axis of propeller shaft rotation. The Y axis is the vertical reference or spindle axis about which a controllable pitch blade is rotated to change pitch. The Y axis is positive up and thus the X axis is positive to starboard.

This axis system was chosen because it maintains the XY plane as the major projection plane when looking forward thus placing the principal transverse view in the most widely used XY coordinate plane. Blade cross sections are described in the XZ plane which is close to the convention used for NACA section data where X is the chordline axis.

2.3 Pitch

As described previously, a propeller blade is formed from fully developed sections located at specified radii R_i placed so that their chords or pitch lines are set at angles ϕ_i to the horizontal X axis (Figure M.4). This angle ϕ, called the pitch angle, follows the right hand convention of positive angles counterclockwise about the Y axis. A view of the section on the XZ plane is also shown in Figure M.5 which represents a surface of a cylinder of radius R_i rolled out to its fully developed length $2\pi R_i$. When this surface is wrapped around a cylinder of radius R_i, the chord line and its extension forms a true helix.

In one revolution of the cylinder, a point on the helix travels a negative Z distance $P_i = 2\pi R_i \tan\phi_i$. The pitch P_i is the same axial distance a screw, of the same cylindrical diameter and pitch angle, would advance in one revolution.

The propeller designer must therefore specify the pitch P_i or the pitch angle ϕ_i for each section at its radius R_i. He must also specify the full diameter D of the propeller. Frequently, this data is supplied as dimensionless values P_i/D called pitch to diameter ratios for each radius R_i. The pitch angles if not given can be extracted from this information.

$$\phi_i = \tan^{-1} \frac{P_i D}{2\pi R_i / D}$$

A pitch diagram can be drawn to show the pitch distribution with respect to the cylindrical radii. The diagram should form a smooth surve when plotted with the pitch P_i as the abscissa and the radii R_i as the ordinate, thus acting as a check of the data and the design.

Pitch is also the parameter for specifying the angle to which a controllable pitch propeller is set. Generally, for this purpose, the pitch angle ϕ_i of the blade section at the radius R_i of 0.7 of the full radius is used and the setting is specified as a P/D ratio where:

$$\frac{P}{D} = \frac{2\pi R_{07}}{D} \tan\phi_{07}$$

2.4 Skew

Blades of simple geometry have the midchords or points of maximum thickness of the blade sections coincident with the blade spindle axis (Figure M.4). Other blades of more sophisticated geometry have their sections displaced along their chordlines from the spindle axis. This displacement is called skew, SK. It is applied to reduce the unsteady hydrodynamic loading on the blade when the propeller is operating in a highly non-uniform wake behind the ship's hull. Skew can be shown on the fully developed unwrapped view of the blade section (Figure M.6) as a linear dimension along the section chord line from the intersection with the Z axis. The skew of the section is referenced to the midchord or point of maximum thickness. Positive skew moves the section around the shaft axis in the flow direction, opposite to the direction of shaft rotation.

Skew can also be shown as a true linear dimension on the diagram of the expanded outline of the blade (Figure M.7). The fully developed sections are laid out flat on the XY plane with their midchords or points or maximum thickness displaced in the direction of the negative X axis a distance equal to the skew. Positive skew is, therefore, measured along the negative X axis and negative skew along the positive X axis.

Skew is sometime specified in angular form as an angle α_i, projected on the XY plane, through which the wrapped section midchord is displaced (Figure M.8). In this case, the linear skew dimension SK_i for the i^{th} section can be obtained by multiplying the projected skew angle α_i (radians) by the radius R_i and dividing by the cosine of the pitch angle ϕ_i, i.e.:

$$SK_i = \frac{R_i \alpha_i}{\text{Cos}\phi_i}$$

2.5 Rake

Rake (Reference [1]) is an aft or forward displacement of the blade section's midchord or maximum thickness point from the XY plane. It is used to increase clearance between the hull and propeller and to increase efficiency by keeping the blades normal to the flow. As shown in Figure M.6, a blade section can be caused to have rake in two ways. In the case of skewed blades, rake is induced when the sections are displaced along the pitch chord line. The Z distance between the midchord or maximum thickness point and the chord line - Z axis intersection point is called the skew induced rake. The conventional method to introduce rake (RK) is to offset each blade section an additional distance in the Z direction from the X axis. The rake distance RK is from the X axis datum (actually the XY plane) to the chord line - Z axis intersection. Offsets aft produce positive rake and offsets forward produce negative rake.

2.6 Warp

When a blade is given skew, it automatically advances the blade sections aft along the helix imposing the skew induced rake on the blade (Figure M.9). If it is desirable to maintain the blade sections in the plane of rotation, then the blade sections are moved back by imposing negative rake on the sections equal to the negative of the skew induced rake. Warp can thus be defined as the angular displacement equal to the projected skew angle of the blade sections measured from the Y axis with any axial displacement such as skew induced rake removed. The linear dimension for warp on the unwrapped view of the blade (Figure M.6) is $WK_i = R_i \alpha_i$.

2.7 Blade Sections

The logical process for determining the geometry of a propeller begins with the selection of the blade section. The designer may invent his own or choose from a variety of proven sections whose hydrodynamic performance have been characterized and recorded.

A blade section (Reference [2]) is initially described in its fully developed or expanded form (see Figure M.10a). In this form, the shape is often described by offsets TF and TB at fractions C of the section chord line length S. The section contour formed by the upper offsets TB measured from the chord or datum line is called the back or suction side of the section. The section contour formed by the lower offsets TF is called the face or pressure side.

The chord line in addition to forming the datum for the section offsets acts as the line by which the section is angled to the horizontal datum, referred to as the pitch angle (Figure M.10). Normally, the face of the section has offsets that are below the datum chord line which are considered to be positive values. When the offsets are all on or above the chord line such that the face is flat or concave, then the chord line is also referred to as the face pitch line (Figure M.11). When the face is convex the face pitch line is established as a straight line tangent to the face, generally at the maximum thickness point, and therefore does not pass through the leading and trailing edges (Figure M.12). If the face pitch or tangent line is not parallel to the chord line then it forms an angle to the nose-tail chord line and cannot be used to measure the section pitch angle. In this case the chord line or a line parallel to it must be used.

2.7.1 Airfoil Section

If airfoil sections are used for marine propeller blades the offsets are given normal to a camber or meanline rather than normal to the chord line (Figure M.13). When blade section data is presented in the format for NACA airfoil sections, the data has to be converted to offsets from the chord line. The basic thickness forms and mean line data for a series of sections can be found in Reference [3].

In addition to the basic form and the mean or camber line data, the lift coefficient and the nose and tail radii are also supplied. For example, a section from the NACA five digit series may be specified as NACA 16-309 with a mean line NACA a = 1.0. This is interpreted as a 16 series basic form with a thickness to chord ratio T/S of .09 or nine percent as indicated by the last two digits. The third digit is the lift coefficient Cl_l expressed as a decimal. The lift coefficient in this example is 0.3. Referring to Figure 16 and page 74 of Reference [3], the mean line curve is plotted from the data table or from an equation such as the following, which is for a uniform chordwise load distribution.

$$y_c = -\frac{Cl_i}{4\pi}\left[\left(1 - \frac{x}{S}\right)\ln\left(1 - \frac{x}{S}\right) + \frac{x}{S}\ln\frac{x}{S}\right]$$

$$\frac{dy_c}{dx} = \frac{Cl_i}{4\pi}\left[\ln\left(1 - \frac{x}{S}\right) - \ln\left(\frac{x}{S}\right)\right]$$

where

x = distance along chord from the leading edge as percent of chord.
y_c = mean line offset ordinate at x as percent of chord.
$\frac{dy_c}{dx}$ = slope of the mean line at x.
S = length of chord line.
Cl_i = lift coefficient

The basic thickness form data, Reference [3], is applied to the mean line ordinates and normal to the mean line local slope dy_c/dx. Other lift coefficients can be accommodated by multiplying the tabulated values of y_c and dy_c/dx by the ratio of the desired lift coefficient to given lift coefficient.

The normals to the mean line are then transformed to offsets from the chord line for the upper (back) and the lower (face) surface lines of the section by the following expressions. The expressions are based on a local axis system that agrees with airfoil convention except that the leading edge is to the right and x is positive to the left (Figure M.9). The expressions for TB and TF, the back and face offsets are introduced at this point as these symbols were used previously for non-airfoil sections. In addition, their use allows the algebraic signs for the offsets to be made consistent with those chosen for the blade axis system where offsets above the chord line are negative and below the chord line are positive. This approach thus allows the data tables from Reference [3] to be used directly to obtain TB and TF values.

$$\theta_i = \tan^{-1}(Cl_r \cdot \frac{dy_c}{dx})$$

$$Cl_r = \frac{Cl_{in}}{Cl_i}$$

$$y_{cl} = y_c \cdot Cl_r$$

$$X_U = (x_c - y_t \sin\theta)\ \frac{S}{100}$$

$$Y_U = (y_{cl} + y_t \cos\theta)\ \frac{S}{100}$$

$$TB = -Y_U$$

$$X_L = (x_c + y_t \sin\theta)\ \frac{S}{100}$$

$$Y_L = (y_{cl} - y_t \cos\theta)\ \frac{S}{100}$$

$$TF = -Y_L$$

where

Cl_{in} = desired lift coefficient
Cl_i = given lift coefficient
Cl_r = lift coefficient ratio
θ_i = angle of the normal to the mean line at x_c value of Cl_i
x_c = chord position from leading edge expressed as percentage of the chord
y_t = basic form offset and length of normal to the mean line at x as percentage of chord
y_{cl} = mean line offset at x_c as percentage of chord
X_U = chord line distance to the point where the normal to the mean line meets back contour of section
Y_U = offset distance from the chord of the section to the point where the normal to the mean line meets the back contour of the section
X_L = chord line distance to where the normal to the mean line meets the face contour of the section

Y_L = offset distance from the chord to a point where the normal to the mean line meets the face contour of the section

TB = offset distance from section chord to back surface of section

TF = offset distance from section chord to front surface of section

When a blade is built up from a single section form, the section is increased in thickness from the top to the blade hub intersection for hydrodynamic and strength requirements. An increase in thickness can be achieved by increasing the chord length while keeping the proportions the same. This will seldom produce the required thickness without an unacceptable increase in chord length. The most convenient method is to change the maximum thickness to chord ratio, T/S, of the section as required and adjust the basic form data to suit.

The expressions for calculating the section offsets when using a varying T/S ratio become

$$X_U = (x_c - \frac{100y_t}{y_{max}} \cdot \frac{T}{S} \sin\theta) \frac{S}{100}$$

$$Y_U = (y_{cl} - \frac{100y_t}{y_{tmax}} \cdot \frac{T}{S} \cos\theta) \frac{S}{100}$$

$$X_L = (x_c - \frac{100y_t}{y_{tmax}} \cdot \frac{T}{S} \sin\theta) \frac{S}{100}$$

$$Y_L = (y_{cl} - \frac{100y_t}{y_{tmax}} \cdot \frac{T}{S} \cos\theta) \frac{S}{100}$$

where $\frac{T}{S}$ = required thickness to chord ratio.

2.7.2 Interpolation of Section Coordinates

Interpolation of additional section coordinates is frequently required. One reason is that unfortunately the upper and lower offsets when converted to the chord line datum from NACA section data do not always originate from the same chord locations. Though in this form they are useful for graphically presenting the geometry of the blade in spatial X,Y,Z cartesian coordinates, they are not suitable for stress analysis particularly when using the finite element method. It is desirable therefore to generate upper and lower section offsets TB, and TF at identical chord locations.

The computer program BLADGM for generating the desired section coordinates uses cubic spline interpolation which, together with a section plotting program, is available to display fully developed blade sections on a Tektronix 4015 graphics display terminal.

2.7.3 Leading Edge

The leading edge radius (LER), Figure (M.14), is generally given with the basic form data. The center of this radius is conventionally determined by using the slope of the mean line at an x value of 0.5 percent of the chord S. The radius specified in the basic form data is the hypotenuse of the triangle (Figure M.14) at the leading edge with an included angle θ. Thus, the coordinates for the center of the leading edge radius are

$$X_r = \mathrm{LER} \cdot \cos\left(\tan^{-1}(Cl_l \cdot \frac{dy_c}{dx})\right)$$

$$Y_r = \mathrm{LER} \cdot \sin\left(\tan^{-1}(Cl_l \cdot \frac{dy_c}{dx})\right)$$

The trailing edge radius if not otherwise specified is given by TER = .01T where T is the maximum thickness of the basic form.

Often insufficient data is supplied to accurately describe the leading edge region of a propeller blade section. There are a number of methods that can be used for generating the curve through the leading edge and the data supplied for the section. One such method depends on estimating the points of tangency of the leading edge arc (Figure M.14) and the curves through the remaining points of the back and face contours. This method is awkward to program and depends on trial and error fitting of the curves using a spline fit.

A better method uses the same basic thickness form data and the leading edge radius LER to fit a smooth curve through the data points and the leading edge. A cubic spline fit was chosen for this method as it is compatible with the thickness form equations given in Reference [3].

The spline fitting technique is based on passing a set of cubic functions through the points using a new cubic in each interval. The equation for the i^{th} interval between points x_i and x_{i+1} is (Reference [4]).

$$y = A_i + B_i (x-x_i) + C_i (x-x_i)^2 + D_i (x-x_i)^3$$

The cubic must fit the two end points of the interval $x_i \le x \le x_{i+1}$ (Figure M.15a). Therefore at $x = x_i$

$$y_i = A_i + B_i (x_i-x_i) + C_i (x_i-x_i)^2 + D_i (x_i-x_i)^3 = A_i$$

Similarly at $x = x_{i+1}$

$$y_{i+1} = A_i + B_i (x_{i+1}-x_i) + C_i (x_{i+1}-x_i)^2 + D_i (x_{i+1}-x_i)^3$$

An additional requirement is that there must be slope compatibility at the end points of adjoining intervals. For the interval $x_i \le x \le x_{i+1}$

$$\text{at } x = x_i, \frac{dy}{dx} = B_i$$

$$\text{at } x = x_{i+1}, \frac{dy}{dx} = B_i + 2C_i (x_{i+1} - x_i) + 3D_i (x_{i+1} - x_i)^2$$

The A_i, B_i, C_i, D_i coefficients can be obtained from the data supplied provided the offsets y_i and the slopes can be found at the end points of the intervals. Unfortunately, at the leading edge ($x_i = x_1 = 0$) (Figure M.15(a)), which is a point on a circular arc, there is an infinite slope which cannot be used to solve for the coefficients. This problem can be overcome by a transformation of the x axis in the interval $x_1 \le x \le x_2$ of $x=\delta^2$ (Reference [5]).

For the transformed coordinate system, the spline equation is

$$y = A_1 + B_1(\delta-\delta_1) + C_1(\delta-\delta_1)^2 + D_1(\delta-\delta_1)^3$$

$$\text{at } \delta = \delta_1 = 0 \qquad y_1 = A_1 = 0$$

$$\frac{dy}{d\delta} = B_1 = \sqrt{2r}$$

where r = specified leading edge radius LER.

$$\text{at } \delta = \delta_2$$

$$y_2 = B_1\delta_2 + C_1\delta_2^2 + D_1\delta_2^3 = A_2$$

$$\frac{dy}{d\delta} = B_1 + 2C_1\delta_2 + 3D_1\delta_2^2 = 2B_2\delta_2$$

As can be seen from Figure M.15 (b), the transformed curve now has less than infinite slope as $\delta = 0$ and A_1, B_1, C_1, D_1 can be obtained in the interval $\delta_1 \le \delta \le \delta_2$.

Therefore, since $\delta = \sqrt{x}$

$$A_1 = 0$$

$$B_1 = \sqrt{2r}$$

$$C_1 = \frac{1}{\sqrt{x_2}}\left[\frac{3A_2}{\sqrt{x_2}} - 2B_2\sqrt{x_2} - 2B_1\right]$$

$$D_1 = \frac{1}{x_2}\left[2B_2\sqrt{x_2} - \frac{2A_2}{\sqrt{x_2}} + B_1\right]$$

where

r = leading edge radius LER

x_2 = end point of interval $x_1 \le x \le x_2$

A_2 = the basic form offset at x_2 (generally at .5% of chord)

B_2 = the slope at the end point x_2

The slope B_2 is obtained from a cubic spline fit of the offsets y_2 to y_n of the basic form data in the intervals x_2 to x_n. The cubic spline fitting equation defining the section from the leading edge over the interval x_1 to x_2 has been programmed and incorporated in the program BLADGM.

When the section has camber, the leading edge will extend beyond the basic form chord station $x_1 = 0$ (Figures M.14 and M.16). Fitting the leading edge to an already cambered section will introduce additional complications than just curve fitting the leading edge to the basic form before camber is added. If not already given, the offset coordinates have to be shifted temporarily to the chord line running through the extreme leading edge point where there is a leading edge radius r. The chord length must be increased by the amount it extends beyond station x_1 to produce the modified chord length S_m which will become the datum for all the chord stations and section offsets. Then the back and face contours are curve fit separately for the contour of the nose. The interpolation stations for identical back and face positions can be input as selected values or generated within the program by the following equation where β is incremented from 0° to 180° according to the required number of stations.

$$x_i = \frac{S_m}{2}(1 - \cos\beta_i)$$

After curve fitting the nose, the offset coordinates are shifted back to the original chord line position.

In addition to curve fitting the section data, BLADGM also formats the data for plotting or for passing the data through to programs for further processing either for the generation of propeller blade geometry or for finite element modelling.

2.8 Expanded Outline

A propeller blade is generally described by first specifying blade sections at a number of radii whose centres are the center of rotation for the propeller (Figure M.7). The fully developed sections are revolved into the XY plane about their chord lines, which are parallel to the X axis, so that the backs of the sections are up and the faces are down. The sections are oriented so that their chord lines remain at a distance Y equal to the radius for their location from the center of rotation. The mid chord, or another datum such as the point of maximum section thickness, is located either on the Y axis or a specified x distance SK called skew from it. If the section coordinates have been specified from the leading edge as fractions of the chord C_i, then they can now be transformed to the new origin as follows (Figure M.17a). The jth chord location using mid chord as datum for the ith blade section is given by

$$X_{ij} = S_i/2 - (C_j S_i + SK_i)$$

or using the point of maximum thickness as datum

$$X_{ij} = S_i \cdot PMT_i - (C_j S_i + SK_i)$$

where

X_{ij} = jth offset station for ith section

S_i = chord length for ith section

C_j = chord fraction for jth offset

PMT_i = point of maximum thickness as new datum for ith section

SK_i = skew for ith section

A further complication can arise if the chord stations C_j are specified as fractions of the dimensions from the PMT datum to the leading and trailing edges (Figure M.17b). For this case, the chord stations are calculated in two steps. From leading edge to PMT

$$X_{ij} = S_i \cdot PMT_i \cdot C_j - SK_i$$

From PMT to trailing edge

$$X_{ij} = -\left(S_i \cdot (1 - PMT_i) \cdot C_j + SK_i \right)$$

where C_j = fraction of chord length from PMT datum to leading and trailing edge.

The sections are now arranged in the proper relationship to the blade datum which is the Y or pitch axis. A continuous curved line drawn through the leading and trailing edges of the sections forms what is called the expanded outline of the blade projected on the XY plane (Figure M.7). This is generally chosen as the first graphic representation of the propeller blade geometry and gives the chords, section offsets, section radii and skew as true lengths in accordance with the scale of the drawing.

2.9 Curve Fitting the Blade Tip

The geometric details of the propeller blade frequently do not give sufficient details of the tip outline curve. The last radius and section supplied is generally about 0.95 of the full radius of the propeller (Figure M.18). It is necessary therefore to complete the blade outline at the tip. This is not a difficult job for the pattern marker or the draftsman as they can fit a curve by eye using drawing instruments. In the case of computer graphics, this process is not practical and another method must be used. The method chosen is similar in most respects to that used for curve fitting the blade section leading edge as described in section 2.6.3 of this report. In this case, the expanded outline is used except that skew is not included. The outline is generated from the end points of the chords. The chord mid points of maximum thickness are located on the spindle or Y axis. Because the tip radius is not specified and is a requirement for the solution, it can be shown (Reference [5]) that, based on a parabolic fit and by using the leading and trailing edge coordinates of the last given section, the radius of curvature at the tip can be given by

$$r = \frac{X_1^2}{2Y_1}$$

where

X_1 = the distance from the leading or trailing edge to the Y axis at the last given section radius.

Y_1 = distance from the last section radius R_l to the full blade tip radius RA.

The tip outline is then generated using the cubic spline equation described in section 2.6.3. For this operation, it is convenient to shift the origin of the axis system to the tip. In the final calculations, the tip coordinates are transformed back to the original blade axis system and the skew, if any, is added. Section offsets, pitch, skew and rake for the additional chord lengths are obtained by interpolating or extrapolating the original data as supplied by the designer.

2.10 Blade Fillets

The transition between blade and hub is made by fitting a fillet tangent to the blade surface and to the hub or palm. The shapes used for propeller hubs and palms are as shown in Figure M.19. The hub shape is generally defined by a cubic spline curve. For variable pitch propellers, the palm is generally of spherical form although other shapes can equally be modelled by a 16-noded patch. Using a spherical shape guarantees a smooth uninterrupted surface at the blade hub regardless of blade pitch angle. The spherical radius chosen for a palm can be the same or larger than the radius of the hub. When the palm surface is represented by a 16-noded patch it is advisable to use a patch larger than the required surface because of errors in the approximation.

The requirement to fit a fillet for smooth transition between the hub and blade for such a variety of hub forms is most easily accomplished by the pattern maker in the case of cast propellers. It is much more difficult to do in the case of blades machined on an N.C. mill. For N.C. milling, all surfaces must be mathematically described so that machine tool paths can be generated.

The proportions of the fillets are based on hydrodynamic and strength requirements, the idea being a fillet of such proportions as to give a constant stress through the transition while meeting hydrodynamic and manufacturing requirements.

One standard for a constant stress fillet proportions in common use is a compound radius 3T and T/3 where T is the blade thickness at its intersection with the hub (Figure M.20). Unfortunately, this fillet form is not suitable for N.C. machining because it is discontinuous in the second derivative at the tangency points of the two radii. This problem has been overcome by the use of cubic splines to produce surfaces which ensure smoothness and continuity through the second derivative.

2.11 Data Verification

Data plots of the blade data can be generated by the BLADGM auxiliary program BLADPL. These plots allow inspection of the supplied tabular data in graphic form so that any discontinuities in the basic data can be detected. Computer generated plots of the following data can be produced:

(a) Pitch angle versus blade section radius (Figure M.21).

(b) Mean line slope versus chord fraction (Figure M.22).

(c) Camber/maximum camber versus chord fraction (Figure M.23).

(d) Skew versus blade section radius (Figure M.23).

(e) Rake versus blade section radius (Figure M.24).

(f) Section chord versus blade section radius (Figure M.25).

(g) Maximum camber versus blade section radius (Figure M.26).

(h) Maximum thickness versus blade section radius (Figure M.27).

(i) Lift coefficient versus blade section radius (Figure M.28).

(j) Individual section plots (Figure M.29).

2.12 Generation of Spatial Coordinates

The spatial coordinates for the blade are generated by transforming the two dimensional coordinates of the expanded view and the blade sections into the three dimensional coordinates of the blade in its final form (Figure M.30). The fully developed sections in the expanded view (Figure M.7) show skew as an offset SK_i of the mid chord from the Y axis. As stated previously, in some cases the point of maximum thickness PMT_i is used rather than the mid chord point. If the blade has rake, the unwrapped sections on the XZ plane are offset a distance along the Z axis an amount equal to the rake RK_i (Figure M.6). Each section is rotated about the point of intersection of the chord line with the Z axis to its specified pitch angle ϕ_i, measured between the pitch chord line and the X axis. Angles measured counterclockwise from the X axis are positive. If there is no rake, the chord line runs through the origin of the XZ axis system which becomes the center of pitch rotation.

The X and Z coordinates of the midchord $S_{i/2}$ are

$$ZK_i = SK_i \sin\phi_i + RK_i$$

$$XK_i = -SK_i \cos\phi_i$$

When warp is specified instead of skew, then

$$ZK_i = 0$$

$$XK_i = R_i\alpha_i = WK_i$$

The shape of the section about the pitch chord line is described in the local coordinate system X'Z' by the thickness offset TB_{ij} and TF_{ij} at fractions C_j of the chord line S_i. The coordinates of the face and back can be transformed from the local system to the global XYZ system using the following equations and remembering that TB_{ij} is negative above the chord line and positive below (see Figure M.30).

$$XB_{ij} = RA_i \sin\phi_{ij}$$

$$YB_{ij} = RA_i \cos\phi_{ij}$$

$$ZB_{ij} = ZK_i + S_i\,(C_j - 1/2)\sin\phi_i + TB_{ij}\cos\phi_i$$

where

$$RA_i = R_i - RR$$

$$\alpha B_{ij} = XD_{ij}/RA_i$$

$$XD_{ij} = XK_i - S_i\,(C_j - 1/2)\cos\phi_i - TB_{ij}\sin\phi_i$$

and

R_i = fraction of full propeller radius RR

C_j = fraction of chord length S_i

RA_i = cylindrical radius for section i

TB_{ij} = back of blade section offset at C_j of radius R_i

ZB_{ij} = Z coordinate of back of blade section offset TB_{ij}

XB_{ij} = X coordinate of back of blade section offset TB_{ij}

YB_{ij} = Y coordinate of back of blade section offset TB_{ij}

ZK_i = Z coordinate of mid chord

XK_i = X coordinate of mid chord

S_i = chord length of section at R_i

XD_{ij} = projected arch length from origin to point TB_{ij}

ϕ_i = pitch angle of chord S_i at R_i, positive when measured counter clockwise from the X axis.

αB_{ij} = projected angle in radians of offset TB_{ij} from the Y axis at R_i

The same equations are used for the face (pressure side) coordinates of the blade except that TF_{ij} are substitutes for TB_{ij}, etc.

In this manner, the two-dimensional data supplied by the designer is converted to three-dimensional cartesian coordinates and projections of the blade on the XY, XZ and YZ planes can be produced.

2.13 Conventional Views of the Blade

2.13.1 Transverse View

Once the propeller blade can be described in three dimensional cartesian coordinates, various views can be produced. One conventional view is a projection of the XY plane which is generally referred to as the transverse view. It is a view of the blade an observer would see if he were standing astern of the ship looking forward at the propeller set at its design pitch (Figure M.31).

2.13.2 Longitudinal View

This is a projection of the blade on the YZ plane. It is a view an observer would see when looking at the propeller from the starboard side (Figure M.32).

2.13.3 Plan View

The plan view is not always given. When supplied, it is a projection of the blade on the XZ plane (Figure M.33).

2.13.4 Pitch Diagram

The pitch diagram, Figure M.34, is drawn to show the pitch variation with blade radius. It should form a smooth curve from which the P/D ratio can be determined. From propeller series design charts, the P/D ratio can be used to determine the propeller efficiency.

2.13.5 Blade Thickness and Rake Diagram

The points on the face of the blade at the maximum section thicknesses are located with reference to the spindle or Y axis which lies in the plane of propeller rotation. A line is drawn through the points representing the face side of the thickness diagram. The maximum thickness for each section is located by measuring forward from the face line a distance equal to the thickness. The line joining these points forms the back side of the thickness diagram (Figure M.35). The rake angle is the angle between the line drawn from the tip to the intersection of the plane of rotation and the shaft axis and the plane of rotation.

2.14 Rotation of Blade Axes

It is useful to have the ability to rotate the propeller blade and geometric axes system to new positions to produce a variety of views which will give a better representation of the blade than straight orthographic projections. Rotations can be about either the global XYZ axes or the local xyz axes system using the same coordinate axis transformation matrices. However, the particular axes system selected will dictate the size of the rotation angles and the order in which the transformation matrices are applied. For example, in accordance with the right hand rule, isometric drawings can be produced on the XY plane either by first rotating -45° about the vertical global Y and then 30° about global X or by rotating -45° about local y, -24.94° about local z and then 24.94° about local x. In the latter case, however, the local axis system rotates with the body and the multiplications of the rotational transformation matrices have to be done in the reverse order.

For rotation about the global Y or local y axis, ϕ, the global coordinates of the end points of the new position of the original xyz propeller axis system are obtained by using the following rotational transformation matrix:

$$[\Phi] = \begin{bmatrix} \cos\phi & 0 & -\sin\phi \\ 0 & 1 & 0 \\ \sin\phi & 0 & \cos\phi \end{bmatrix}$$

Assuming the local axes are unit vectors, the new global coordinates are simply

$$[\{X\}\{Y\}\{Z\}] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix} [\Phi]$$

The same transformation matrix can be applied to any single point. With existing xyz coordinates, the new coordinates of the point would be

$$\begin{Bmatrix} X \\ Y \\ Z \end{Bmatrix}^T = \begin{Bmatrix} x \\ y \\ z \end{Bmatrix}^T [\Phi]$$

For rotation about the global Z or local x axis, α, the coordinate transformation matrix is

$$[\alpha] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 \\ -\sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

For rotation about the global X or local x axis, θ, the coordinate transformation matrix is

$$[\theta] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & \sin\theta \\ 0 & -\sin\theta & \cos\theta \end{bmatrix}$$

As mentioned, the rotations can be combined and the resulting coordinates of the geometry referenced to the global axis position. The sequence of rotations, however, is important when rotating about the local axes because the coordinates are obtained by working back from the final axes position. Thus, for rotations about the y, z and x axes, the transformation matrices are multiplied in the order of $[\theta]$, $[\alpha]$ and $[\phi]$. The net multiplication $[\theta]\,[\alpha]\,[\phi]$ of the three transformation matrices is sometimes referred to as the literal matrix $[L] = [\theta]\,[\alpha]\,[\phi]$.

3. Coordinates for Manufacture and Measurement

The coordinates that are generally generated are for the propeller blade at the design pitch angle. The pitch angle at the 70% radius is used as the datum for the pitch settings when a variable pitch blade is rotated about its spindle axis. At the design pitch setting, the coordinates can be used for stress analysis and graphic representation of the blade. The design pitch is not, however, the best angle for machining with a numerically controlled milling machine or for measuring the blade surface contours. For these two processes, it is best to present the maximum projected area possible to reduce machine tool and measurement gauge travel. Zero pitch at the 70% radius gives close to the maximum projected area for most blades. Other angles may be used if they are found to be more suitable.

3.1 Cartesian Coordinates

To present the face of the blade for measurement or machining, it is rotated about is spindle axis through the angle ϕ to approximately zero pitch. Right handed blades, i.e., blades which advance when the propeller shaft is rotated clockwise, are rotated clockwise about their spindle axis. Left hand blades are rotated counterclockwise. The rotation ϕ requires a coordinate transformation of the same form as was described in section 2.14 (for rotation about the global Y or local y axis). The values of YB and YF are not changed by the rotation. The values of XF, XB and ZF and ZB, however, are modified as follows

$$XMF_{ij} = XF_{ij} \cos\phi + ZF_{ij} \sin\phi$$

$$ZMF_{ij} = XF_{ij} \sin\phi + ZF_{ij} \cos\phi$$

To gain access to the back surface, the blade must be rotated an additional 180°. Thus,

$$XMB_{ij} = XB_{ij} \cos(\phi+\pi) + ZB_{ij} \sin(\phi+\pi)$$

$$ZMB_{ij} = -XB_{ij} \sin(\phi+\pi) + ZB_{ij} \cos(\phi+\pi)$$

The coordinates of the blade are now in a form that permits easy measurement of the blade front and back surfaces provided the measurement device can measure accurately along three axes, X, Y and Z. An N.C. milling machine or a jig borer are devices that can be used for this purpose.

3.2 Cylindrical Coordinates

The blade to this point has been described in spatial cartesian coordinates. However, cylindrical coordinates are better suited for measurement by some types of gauges. For this purpose, the coordinates are converted to an angle measured from the Y axis, a radius R from the center of rotation of the shaft of the propeller, and a Z axis dimension for every section offset given in the designer's original data. The measurement angles are:

Face Angle:

$$\theta_{Fij} = \arc\tan(XMF_{ij}/YF_{ij})$$

Back Angle:

$$\theta_{Bij} = \arc\tan(XMB_{ij}/YB_{ij})$$

The radii are

Face Radii:

$$RMF_{ij} = \sqrt{XMF_{ij}^2 + YF_{ij}^2}$$

Back Radii:

$$RMB_{ij} = \sqrt{XMB_{ij}^2 + YP_{ij}^2}$$

The measurement of the Z coordinates can be more complex if a dial gauge is used and negative values cannot be read directly. If negative values occur in the column of Z offsets, an origin shift is required to convert all the offsets to positive values. This is accomplished by subtracting the largest negative Z from all the others. The subroutine RXYZ of program BLADGM carries out the origin shift when required and prints out the modified Z offsets as a data column titled "GAGE".

4. References

1. Saunders, Harold, Hydrodynamics In Ship Design, Vol. 1, SNAME 461-2, 1957.

2. O'Brien, T.P., The Design of Marine Screw Propeller, Hutchinson & Co. Ltd., London, 1962.

3. Abbott, H.I. and Von Doenhoff, Theory of Wind Sections, Dover Publications, Inc., New York, 1959.

4. Gerald, C.F., and P.O. Wheatley, Applied Numerical Analysis, Addison-Wesley Publishing Co. Massachusetts.

5. Smith, D.R., and J.E. Slater, The Geometry of Marine Propellers, DREA Tech. Memo, to be published.

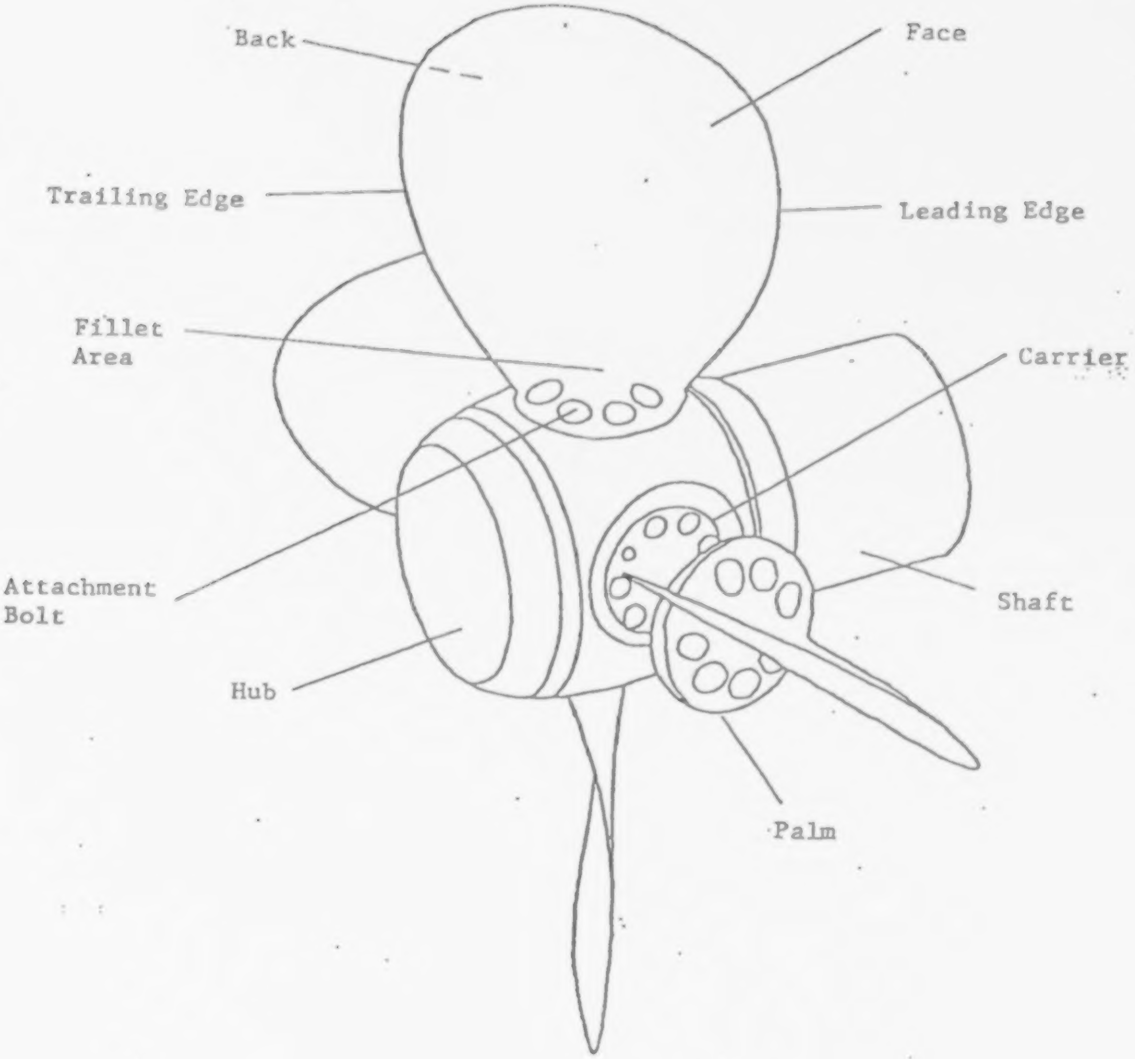

Back
Face
Trailing Edge
Leading Edge
Fillet
Area
Carrier
Attachment
Bolt
Shaft
Hub
Palm

FIGURE M.1: Parts of a Marine Propeller

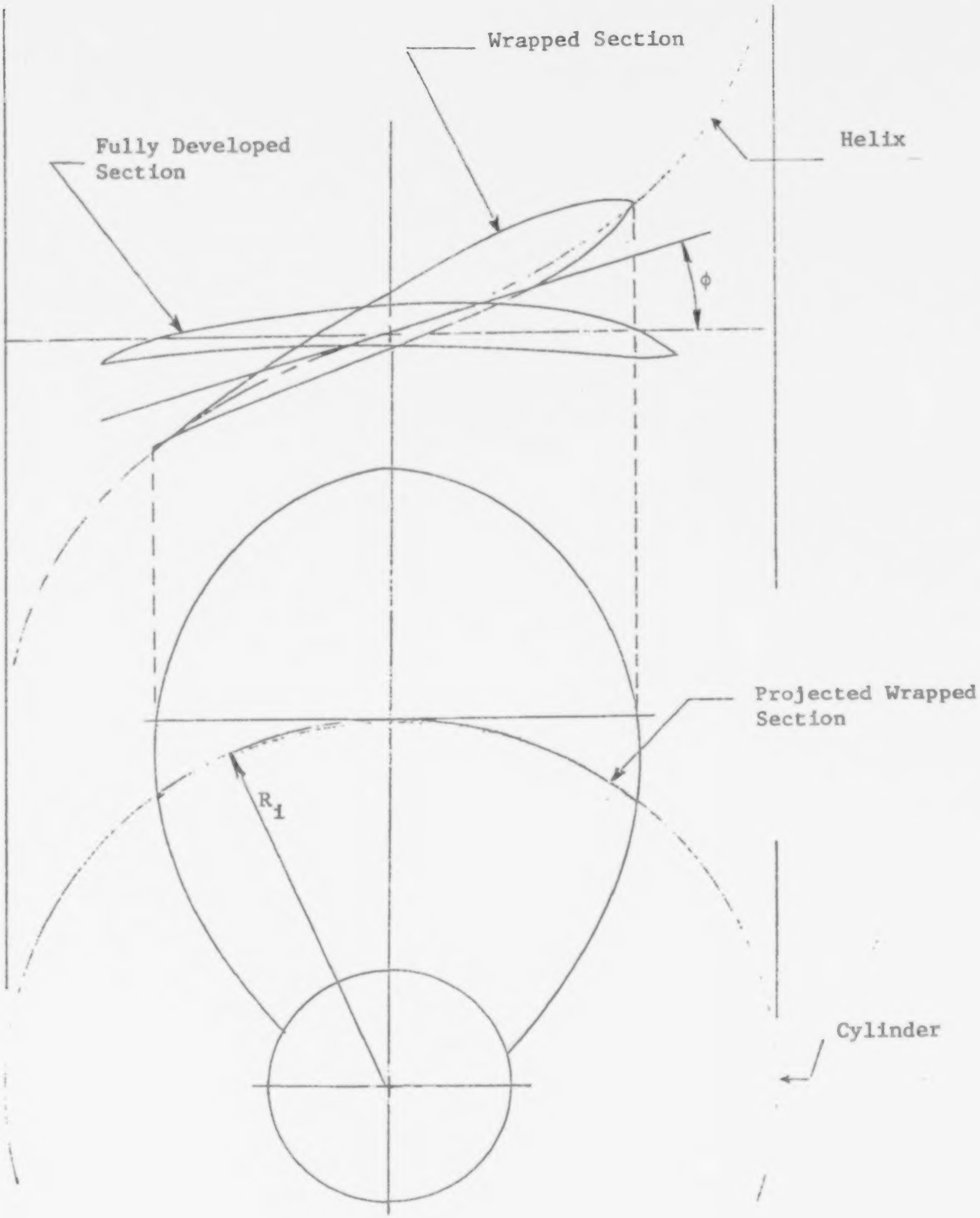
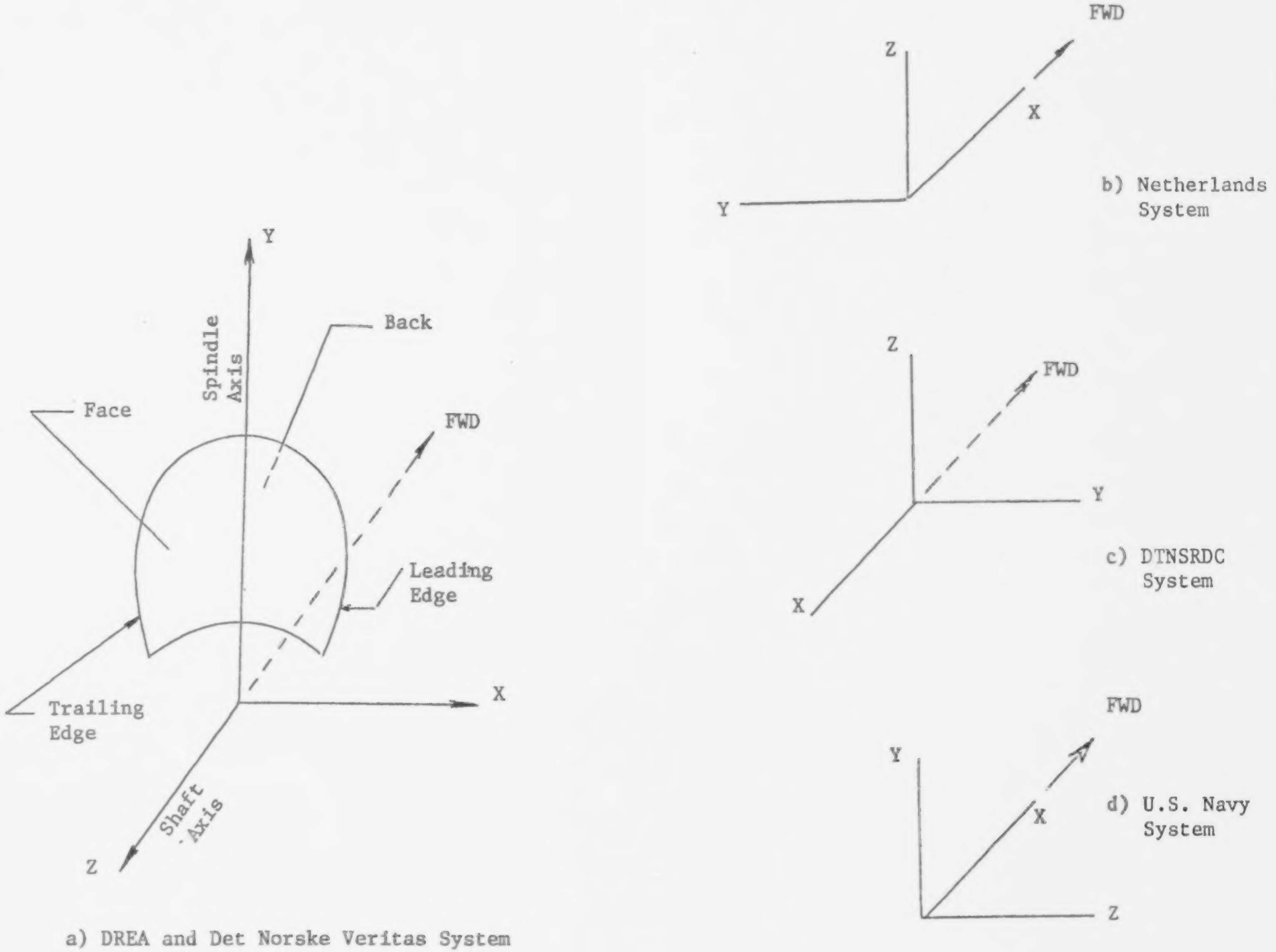

Wrapped Section
Fully Developed Section
Helix
ϕ
Projected Wrapped Section
R_1
Cylinder

FIGURE M.2: Rotating and Wrapping of Fully Developed Sections


Y
Spindle Axis
Back
Face
FWD
Leading Edge
Trailing Edge
X
Shaft Axis
Z
FWD
Z
X
Y
Z
FWD
Y
X
FWD
Y
X
Z

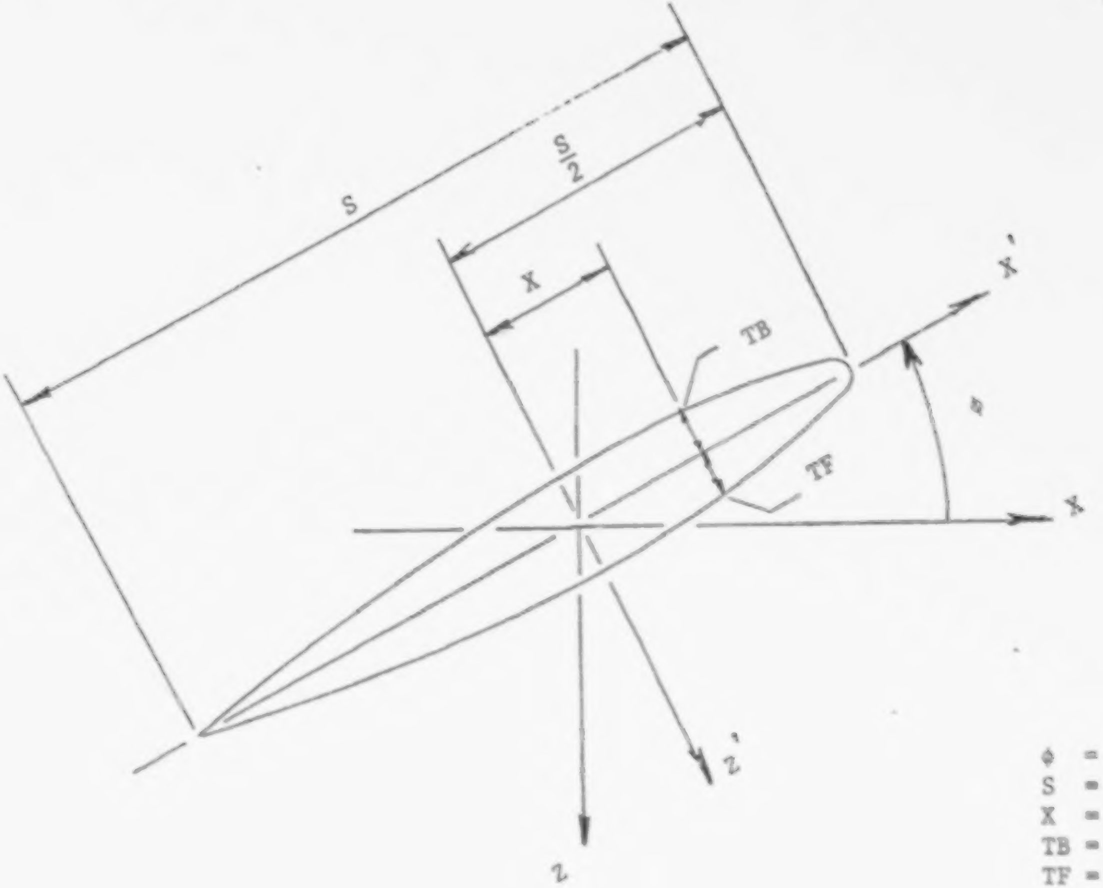
a) DREA and Det Norske Veritas System

b) Netherlands System

c) DTNSRDC System

d) U.S. Navy System

FIGURE M.3: Propeller Axes Systems



ϕ = Pitch Angle
S = Developed Chord Length
X = Chord Station
TB = Offset to Back of Blade
TF = Offset to Face of Blade

FIGURE M.4: Fully Developed Section Projected on XZ Plane with Origin Shifted to Midchord .

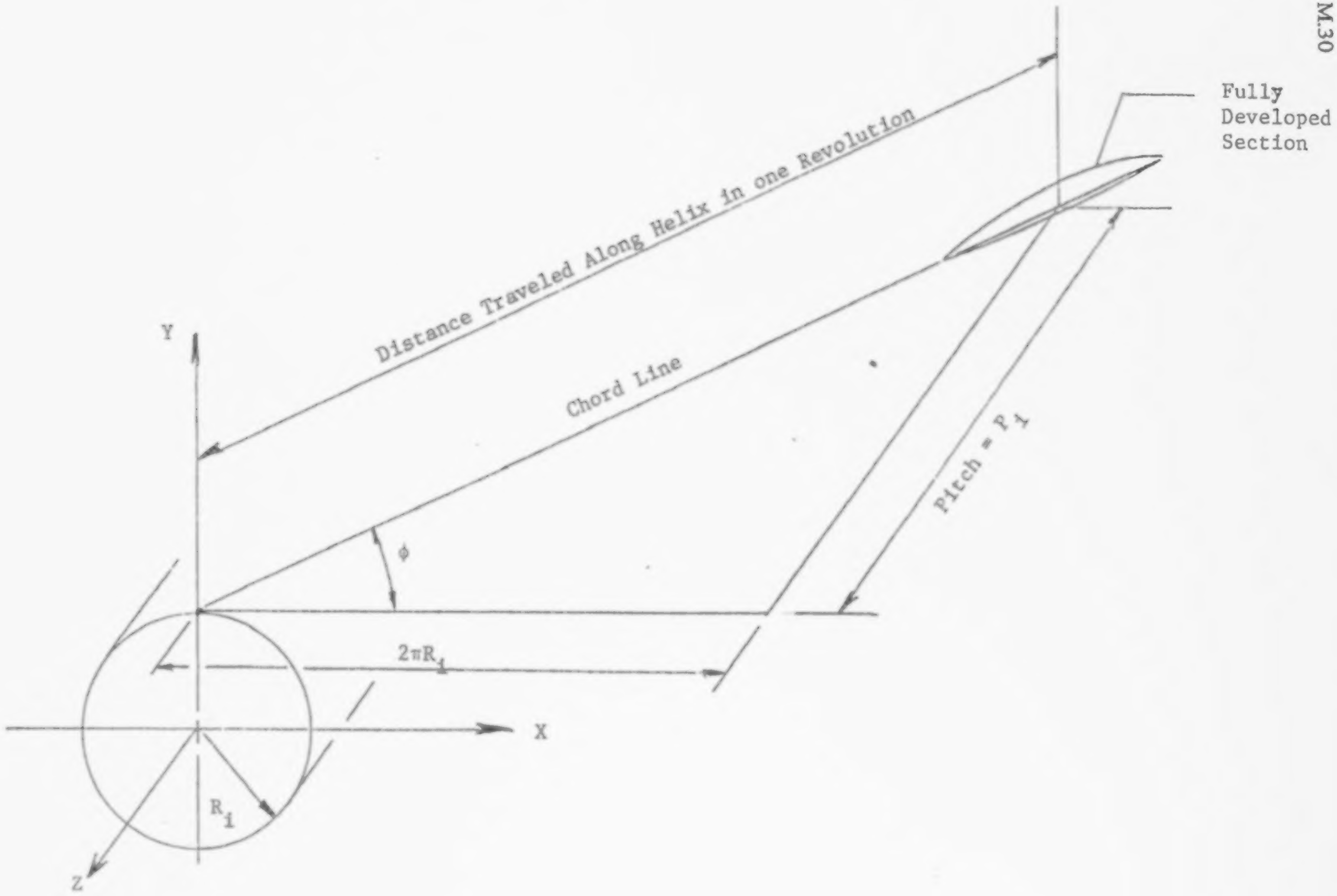

Fully
Developed
Section
Distance Traveled Along Helix in one Revolution
Chord Line
Pitch = P_1
ϕ
$2\pi R_1$
Y
X
Z
R_1

FIGURE M.5: Pitch of an Unwrapped Blade Section by Development of an XY Plane from Cylindrical Surface

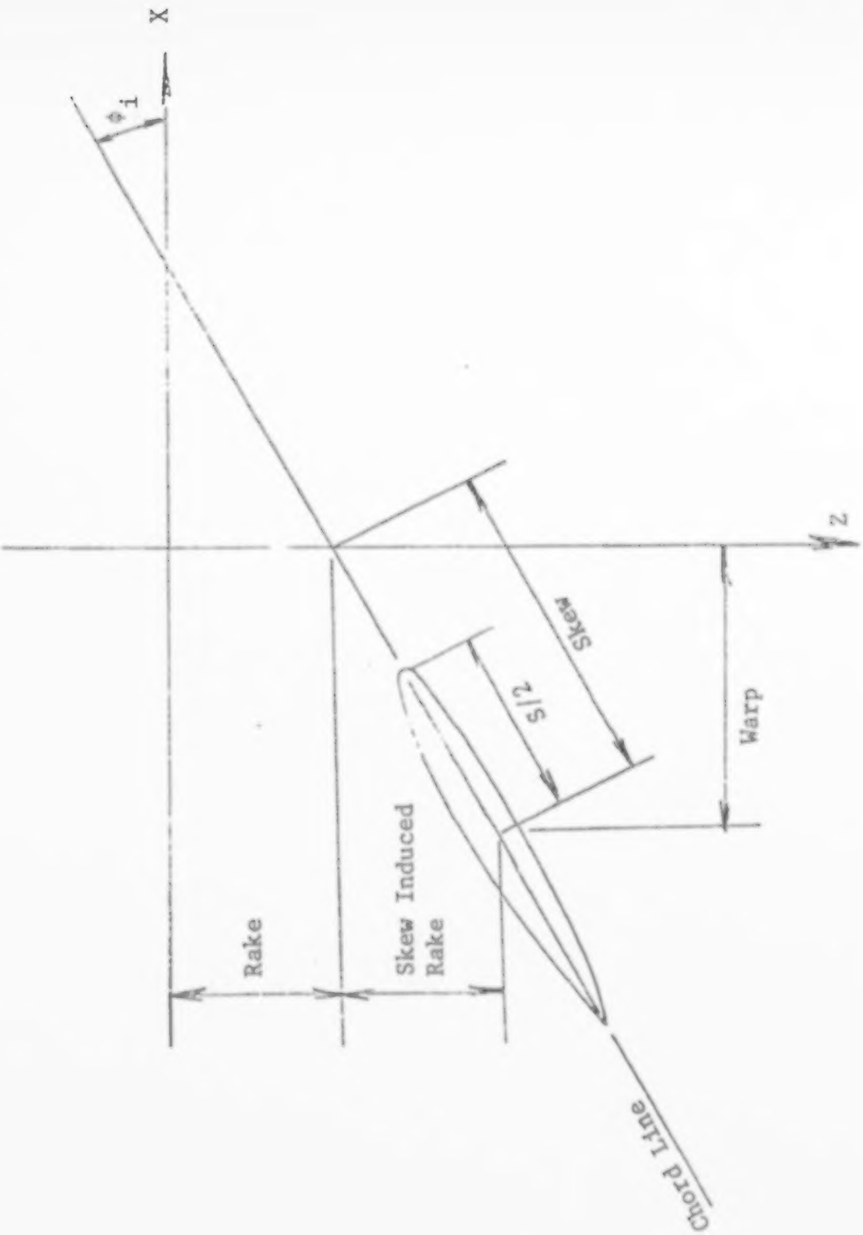

X
ϕ_i
Z
Skew
S/2
Warp
Rake
Skew Induced Rake
Chord Line

FIGURE M.6: Fully Developed Section on XZ Plane Showing Shew and Rake

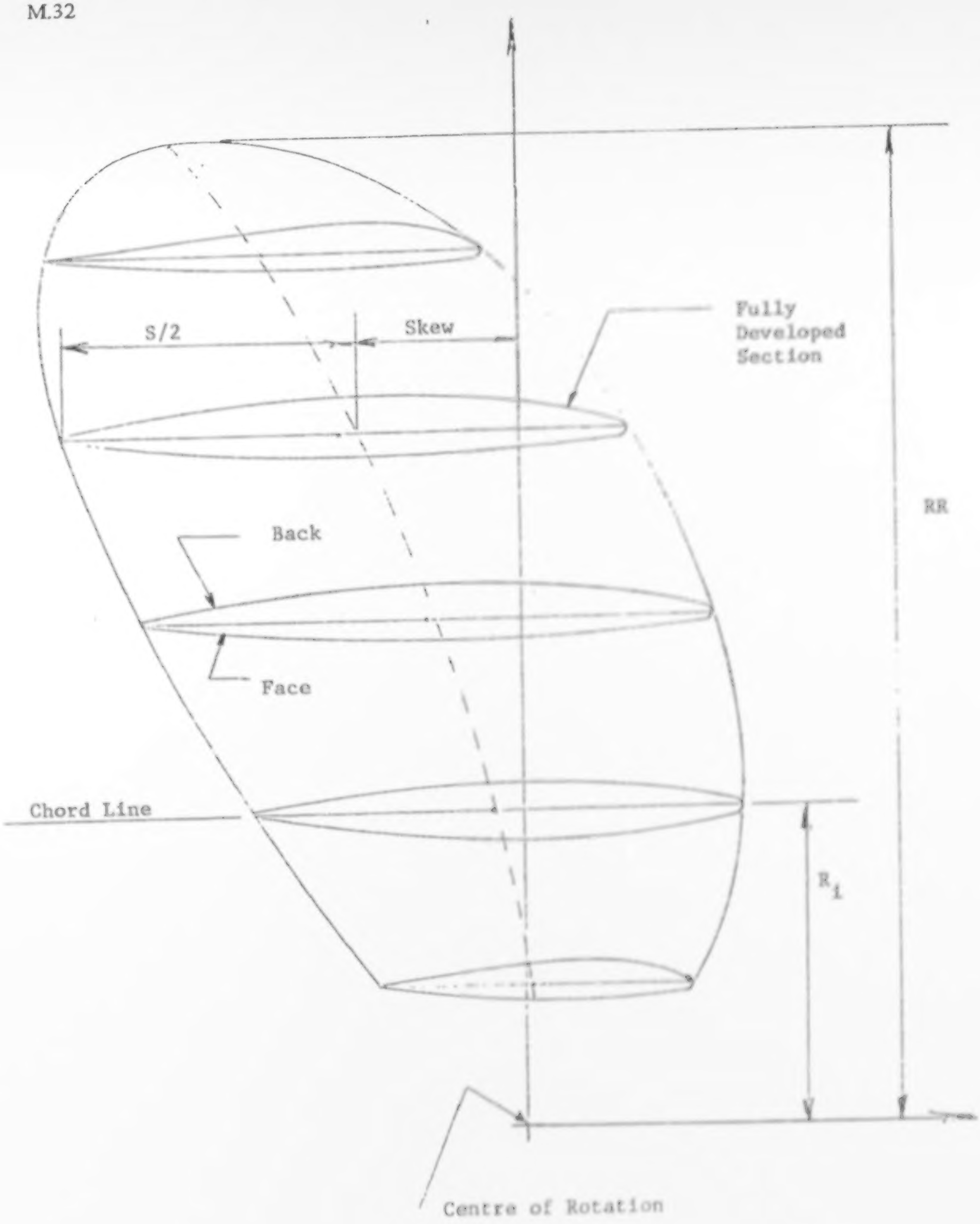

S/2
Skew
Fully Developed Section
RR
Back
Face
Chord Line
R_i
Centre of Rotation

FIGURE M.7: Expanded View of Blade

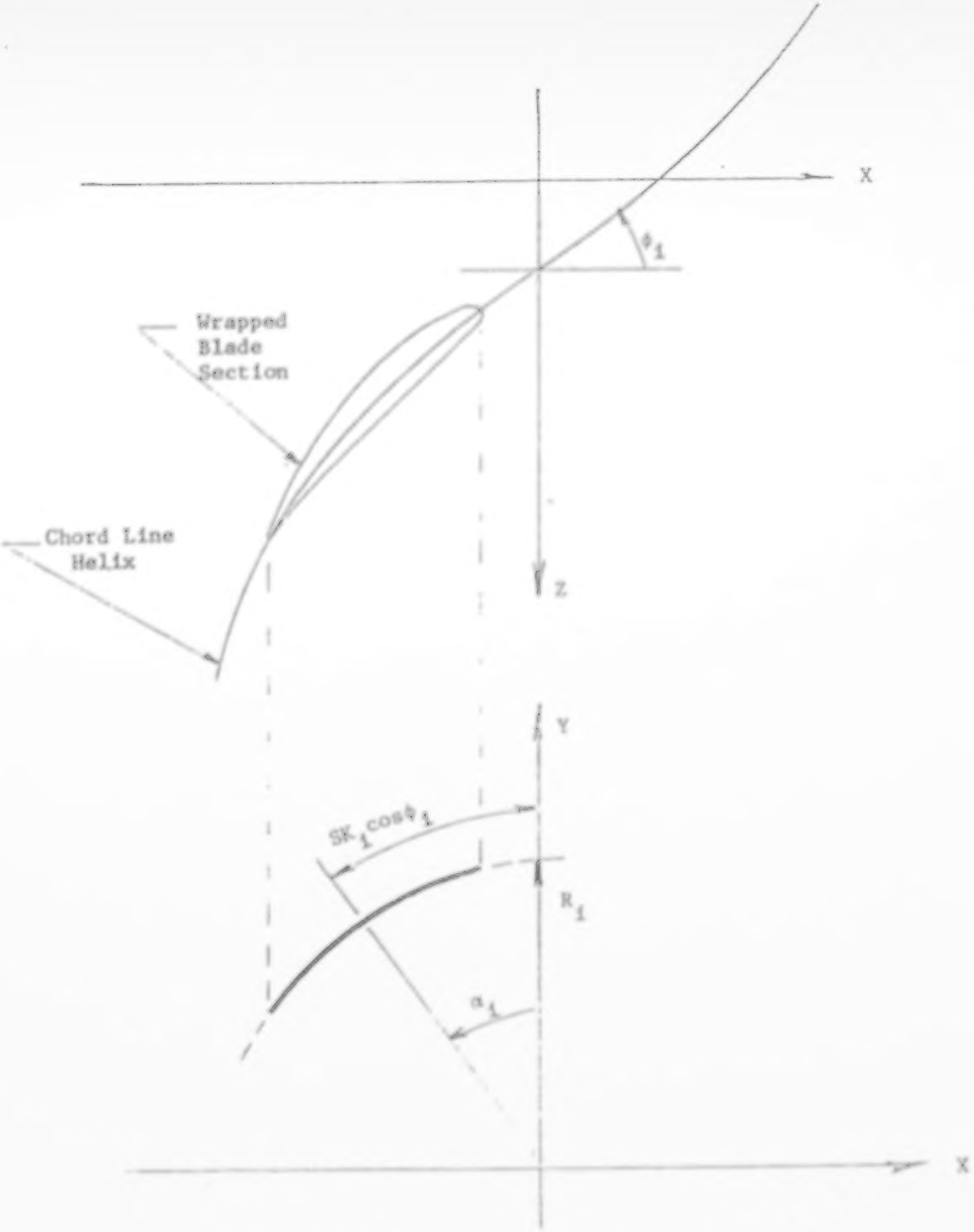

X
ϕ_1
Wrapped Blade Section
Chord Line Helix
Z
Y
$SK_1 \cos\phi_1$
R_1
α_1
X

FIGURE M.8: Wrapped Blade Section Showing Skew Angle α_1

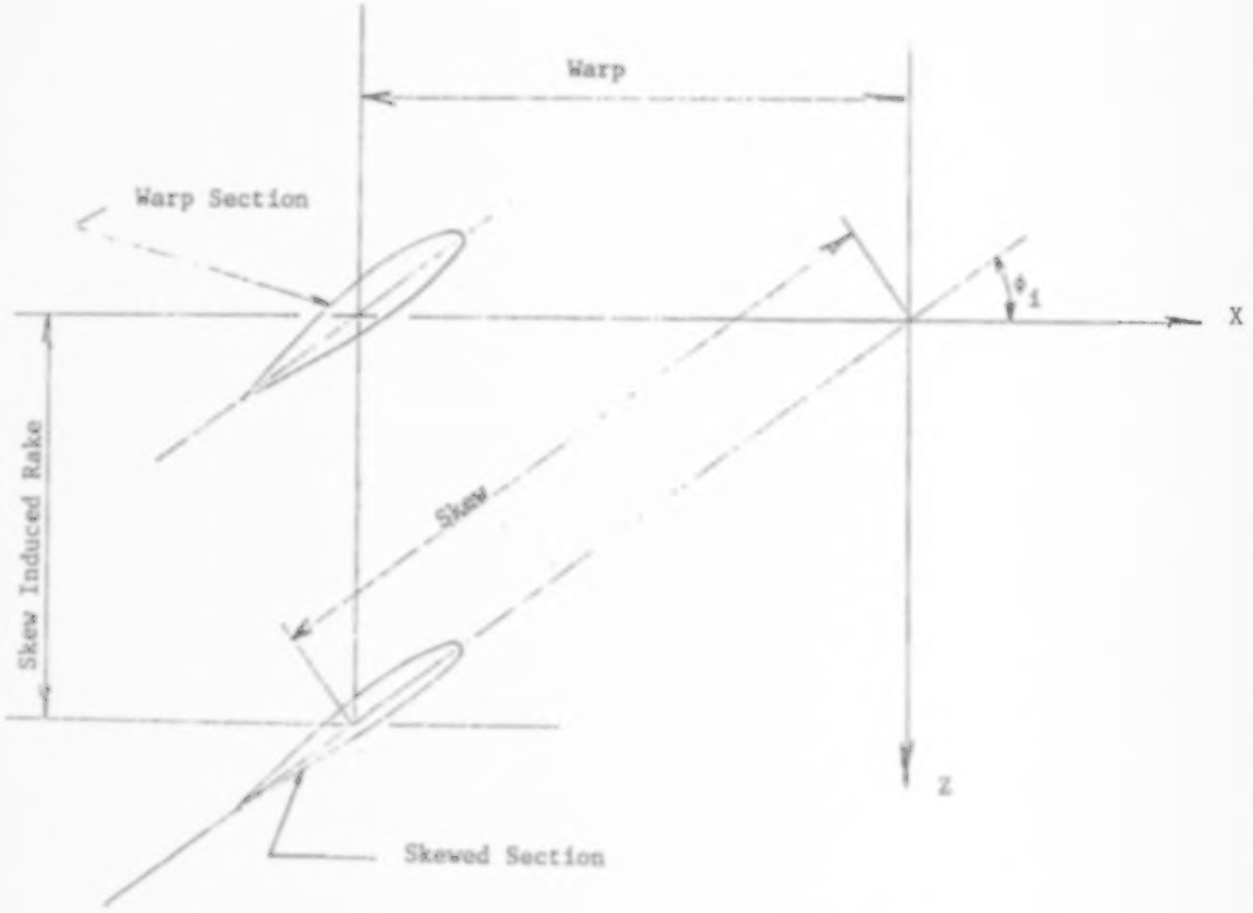
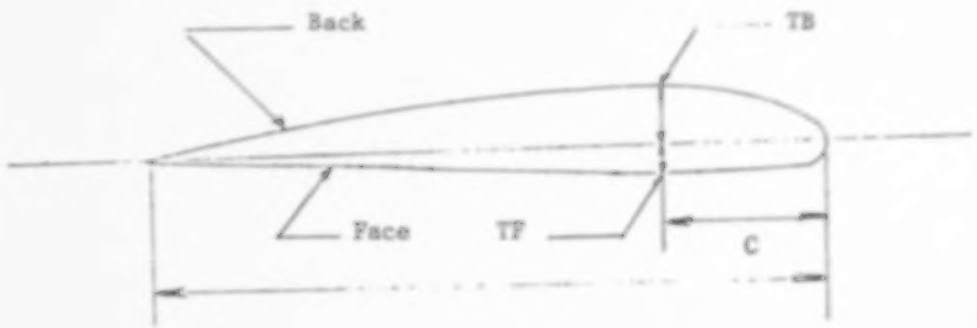
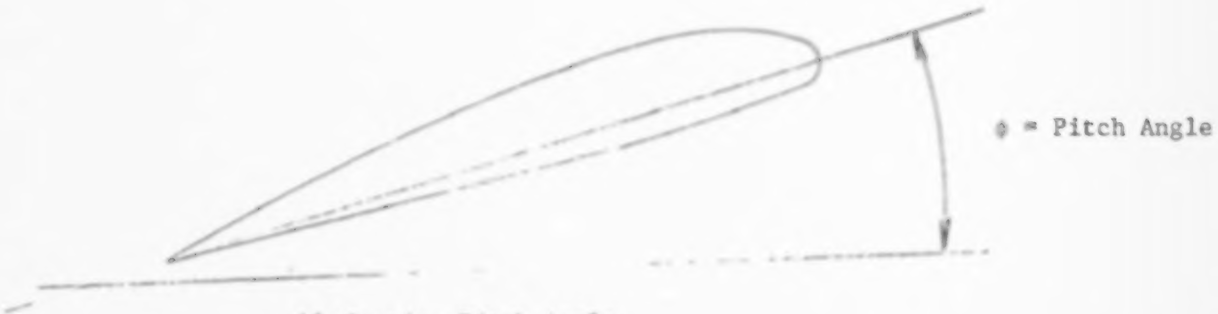

Warp
Warp Section
ϕ_1
X
Skew
Skew Induced Rake
Z
Skewed Section

FIGURE M.9: Propeller Section Warped in the Plane of Rotation


Back
TB
Face
TF
C

a) Expanded Blade Section


ϕ = Pitch Angle

b) Section Pitch Angle

FIGURE M.10: Initial Description of Blade Section

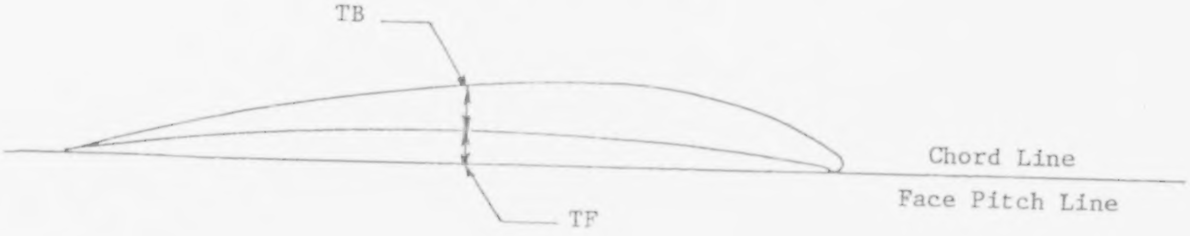

TB
Chord Line
Face Pitch Line
TF

FIGURE M.11: Face Pitch Line for Flat or Concave Sections

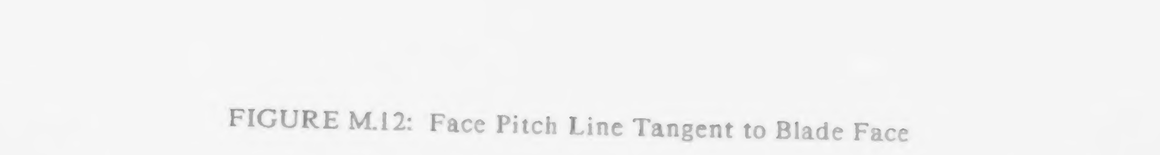

TB
Chord Line
Face Pitch Line
TF

FIGURE M.12: Face Pitch Line Tangent to Blade Face

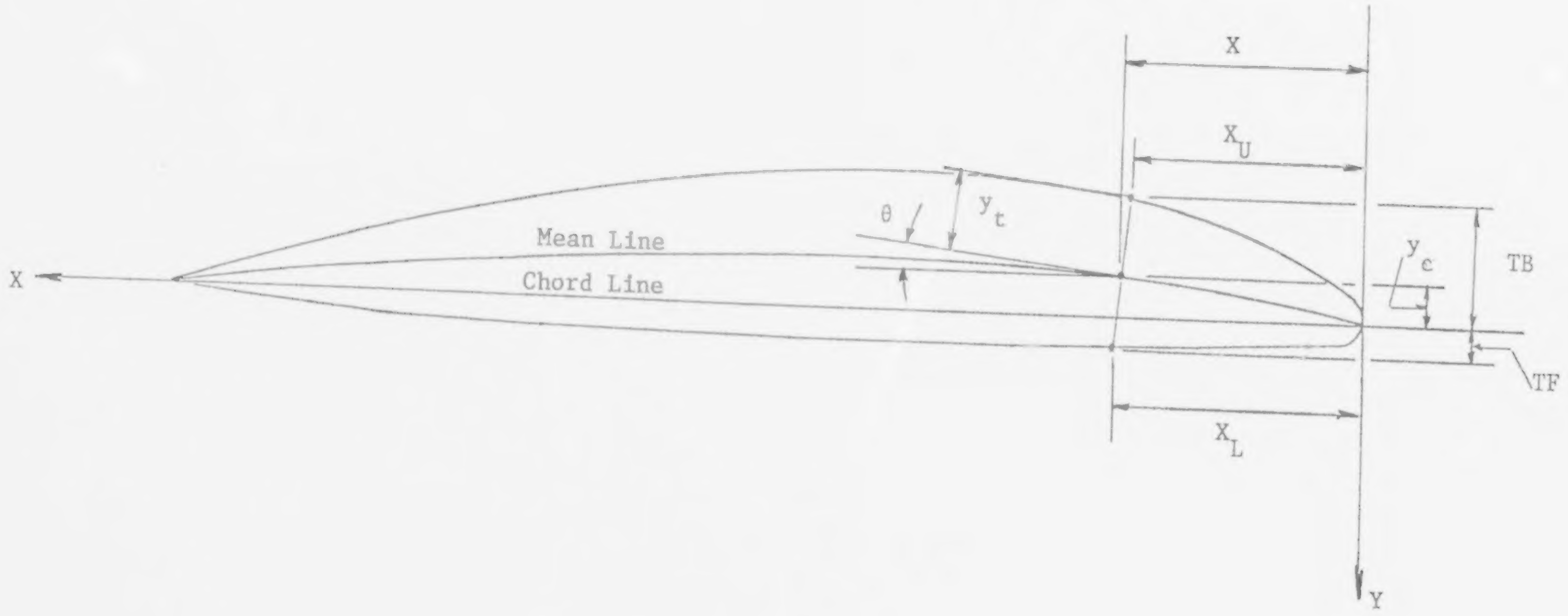

X
X_U
θ
y_t
Mean Line
Chord Line
X
y_c
TB
TF
X_L
Y

FIGURE M.13: Conversion of Mean Line and Basic Form Data to Section Offsets

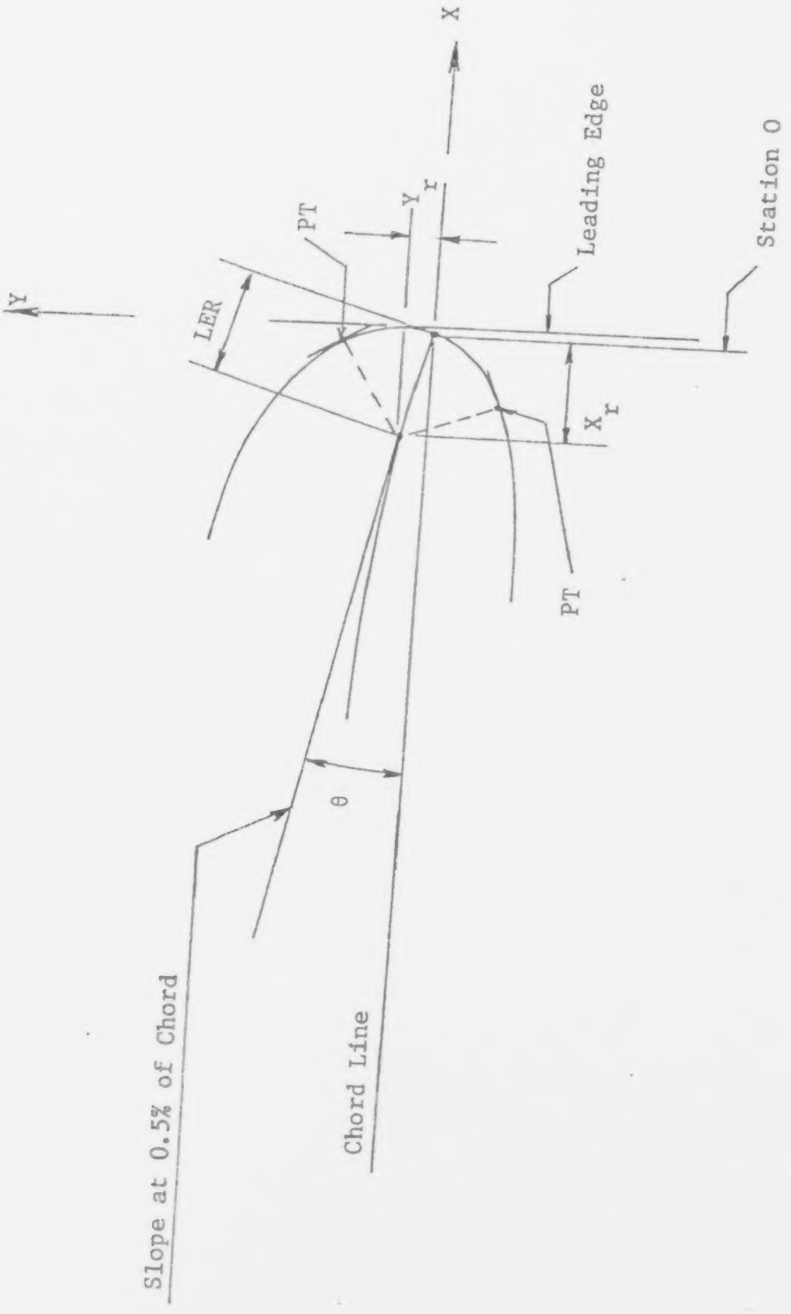
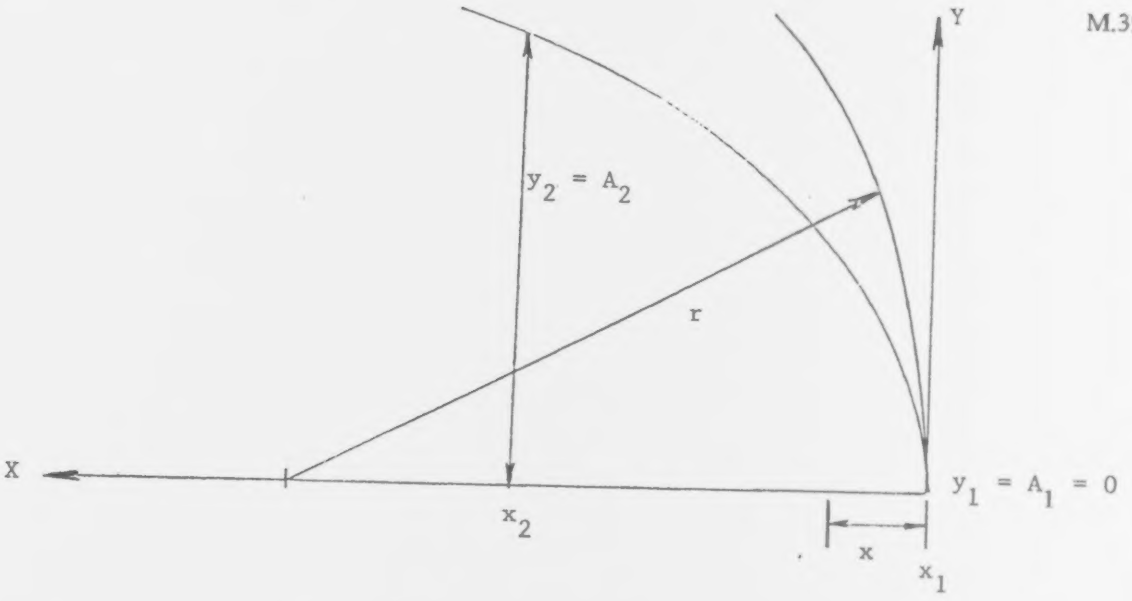
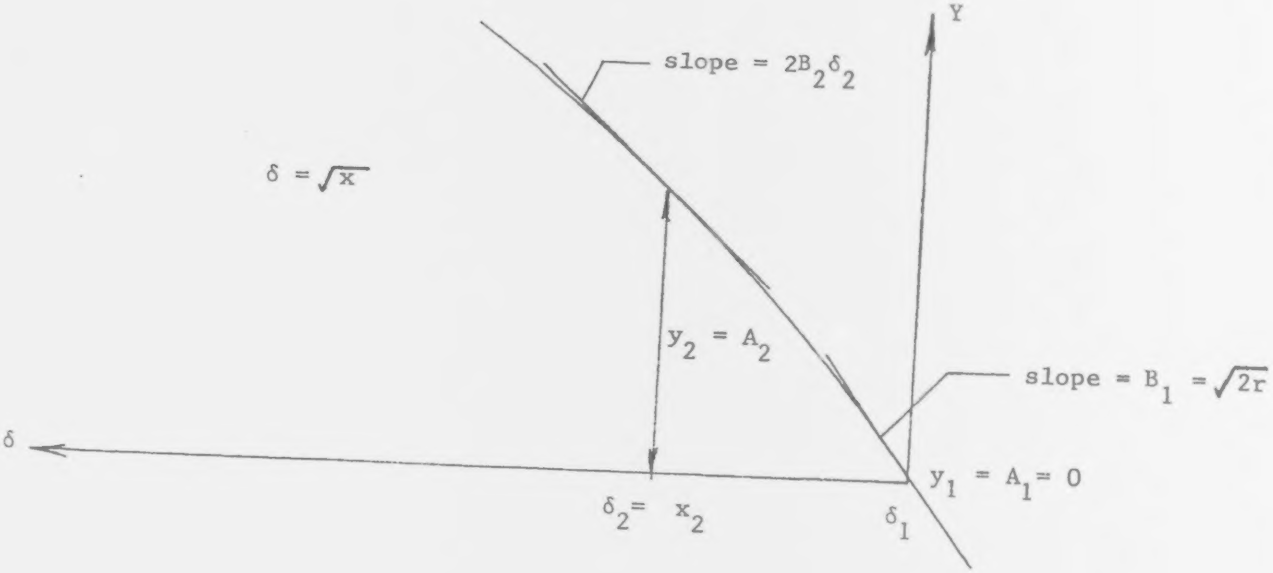

X
Y
Y_r
PT
LER
Leading Edge
Station 0
X_r
PT
θ
Slope at 0.5% of Chord
Chord Line

FIGURE M.14: Leading Edge Geometry


Y
$y_2 = A_2$
r
X
x_2
x
x_1
$y_1 = A_1 = 0$

a) Curve Fitting Intervals for Leading Edge Fit


Y
slope = $2B_2\delta_2$
$\delta = \sqrt{x}$
$y_2 = A_2$
slope = $B_1 = \sqrt{2r}$
δ
$y_1 = A_1 = 0$
$\delta_2 = x_2$
δ_1

b) Transformation of x-Axis to δ-Axis

FIGURE M.15: Spline Fitting Technique for Defining Leading Edge

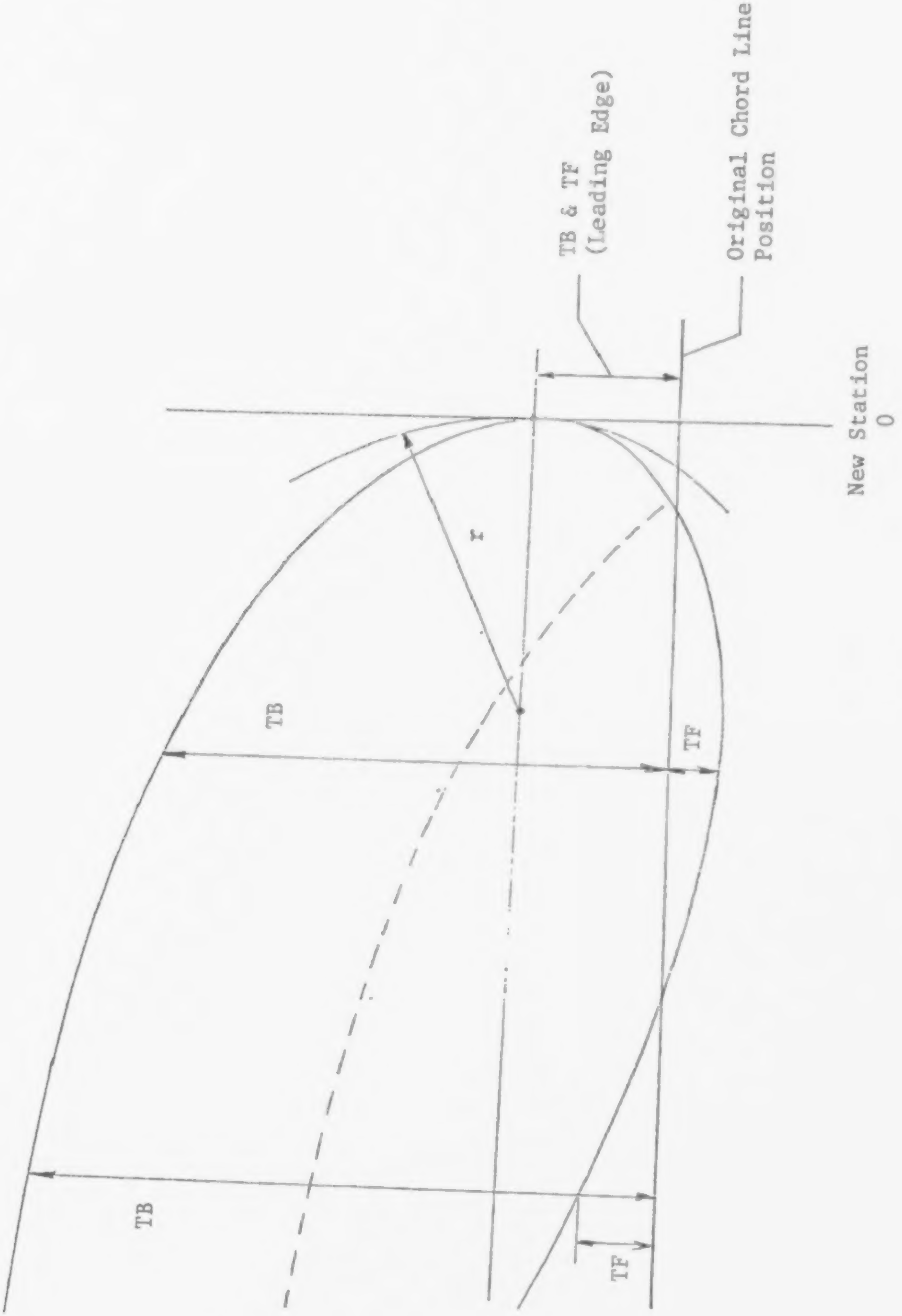
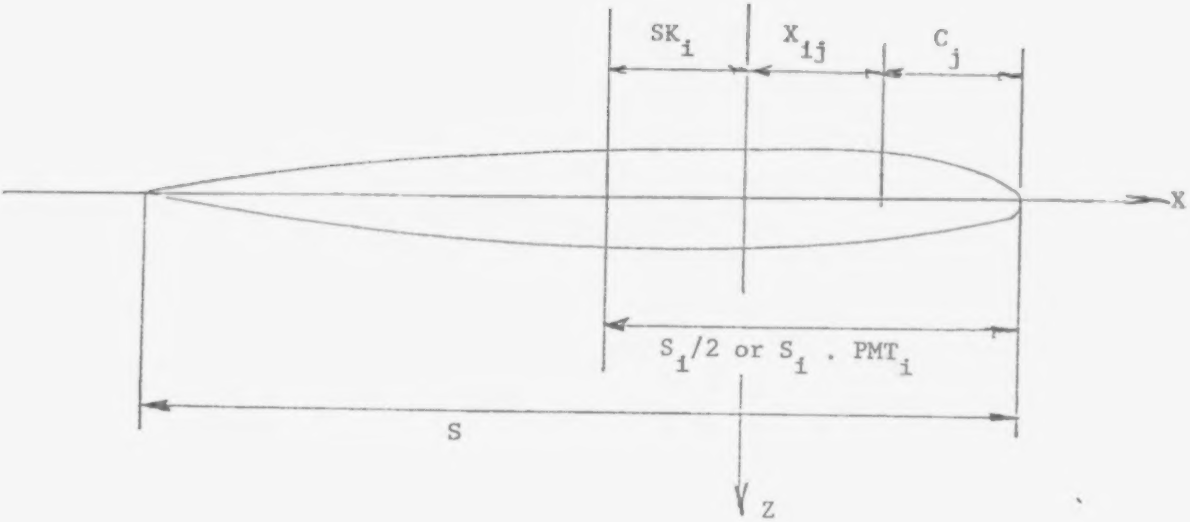
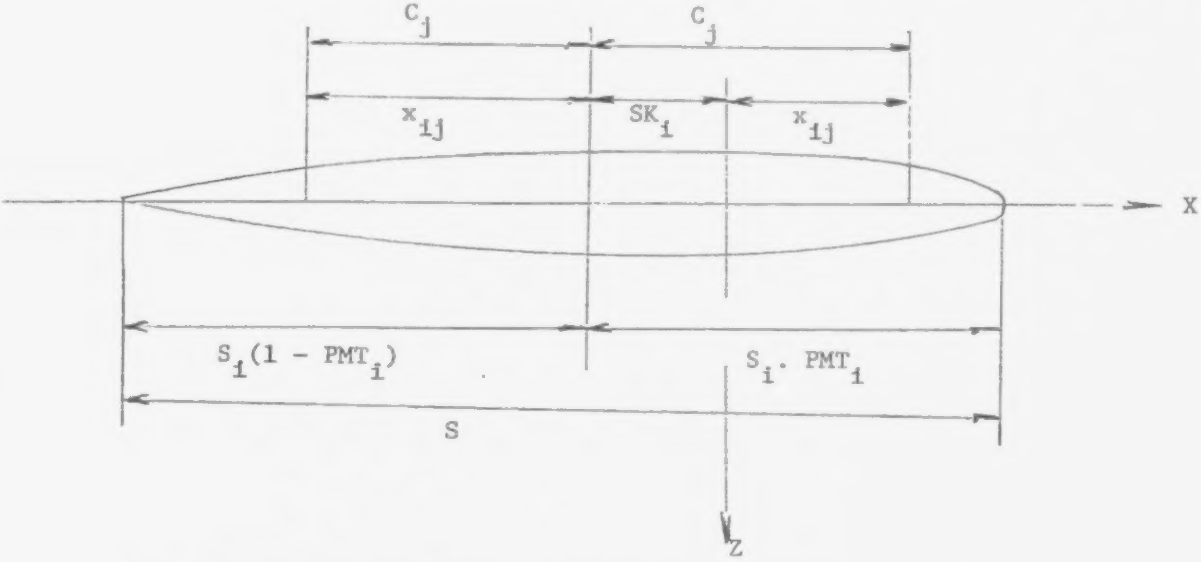

TB & TF
(Leading Edge)
Original Chord Line
Position
New Station
0
r
TB
TF
TB
TF

FIGURE M.16: Leading Edge Radius Fitted to Cambered Section Data


SK_i
X_ij
C_j
X
S_i/2 or S_i . PMT_i
S
Z

a) Chord Stations Specified as Fractions from Leading Edge


C_j
C_j
x_ij
SK_i
x_ij
X
S_i(1 - PMT_i)
S_i. PMT_i
S
Z

b) Chord Stations Specified as Fractions from Point of Maximum Thickness

FIGURE M.17: Transformation of Section Coordinates using Chord Fractions

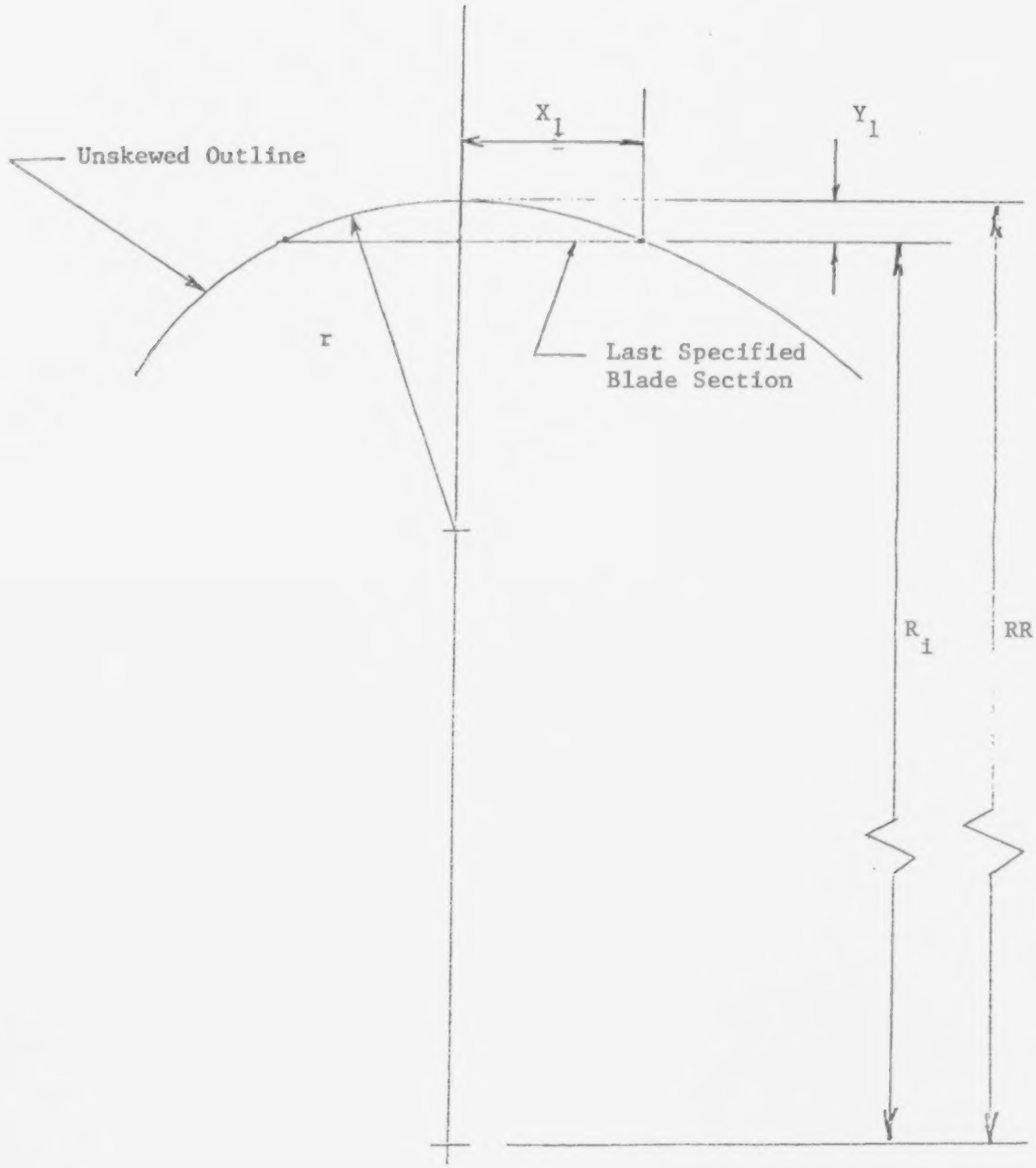

X1
Y1
Unskewed Outline
r
Last Specified Blade Section
Ri
RR

FIGURE M.18: Curve Fitting Blade Tip

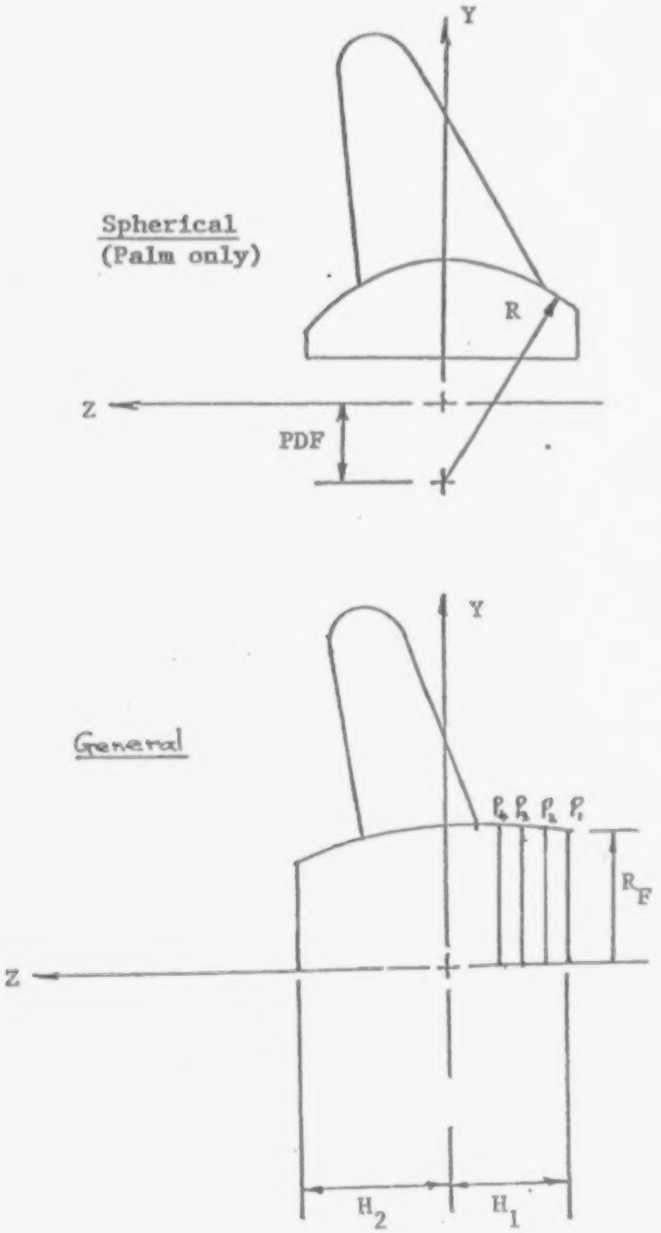

Y
Spherical
(Palm only)
R
Z
PDF
Y
General
P_4 P_3 P_2 P_1
R_F
Z
H_2
H_1

FIGURE M.19: Types of Blades Palm/Hub Models

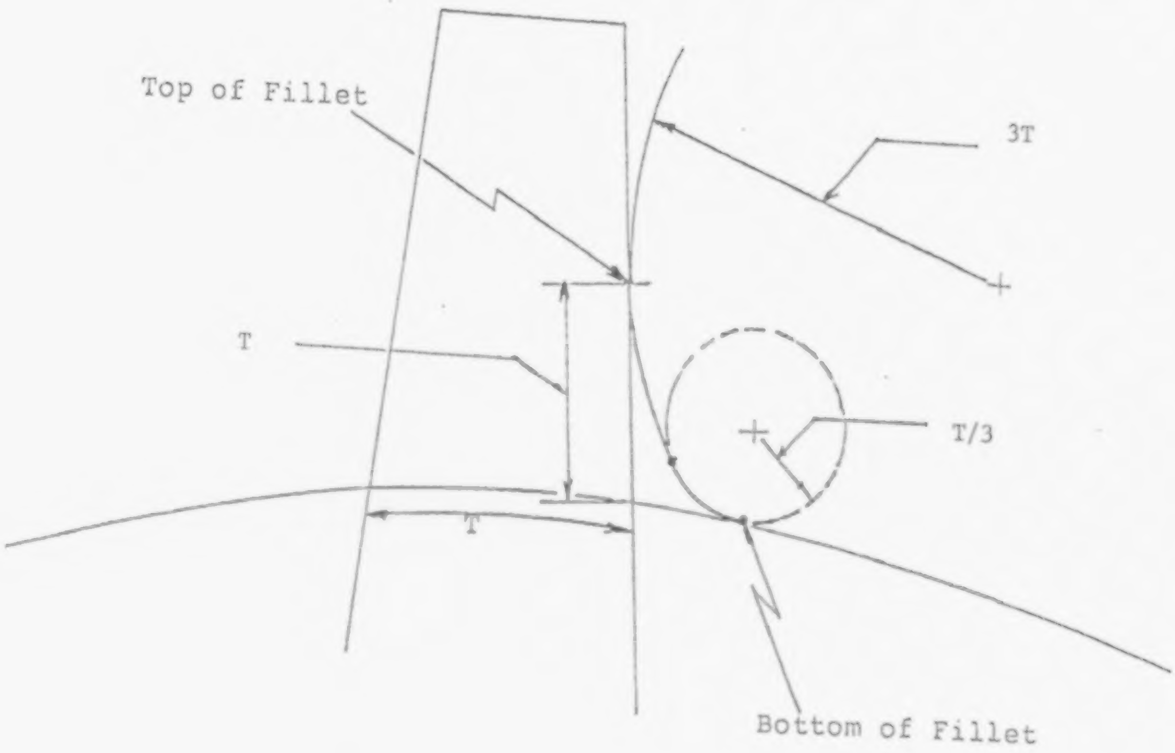

Top of Fillet
3T
T
T/3
T
Bottom of Fillet

FIGURE M.20: Compound Radius Type Fillet

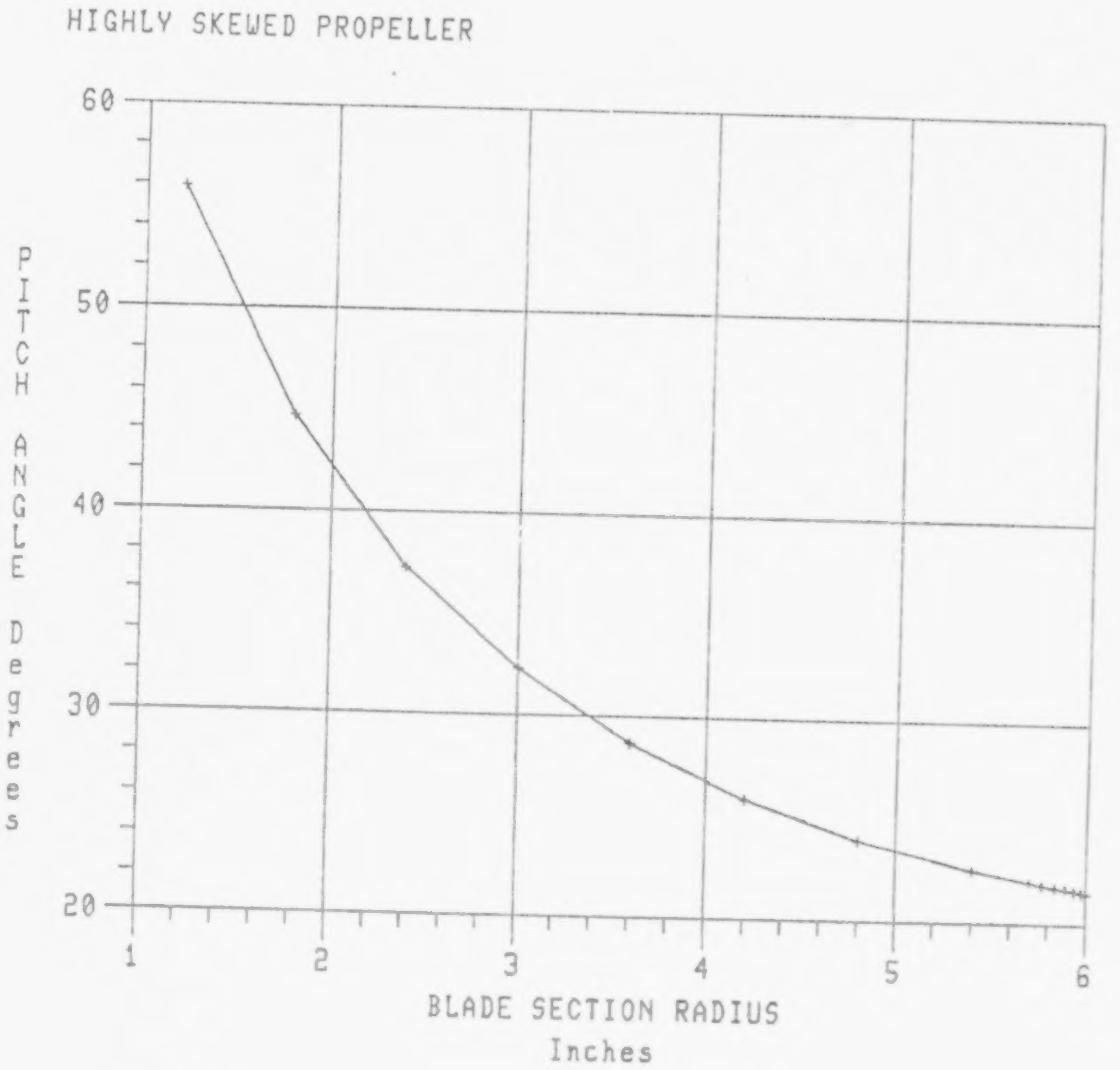


FIGURE M.21: Computer Plot of Pitch Angle Versus Blade Section Radius

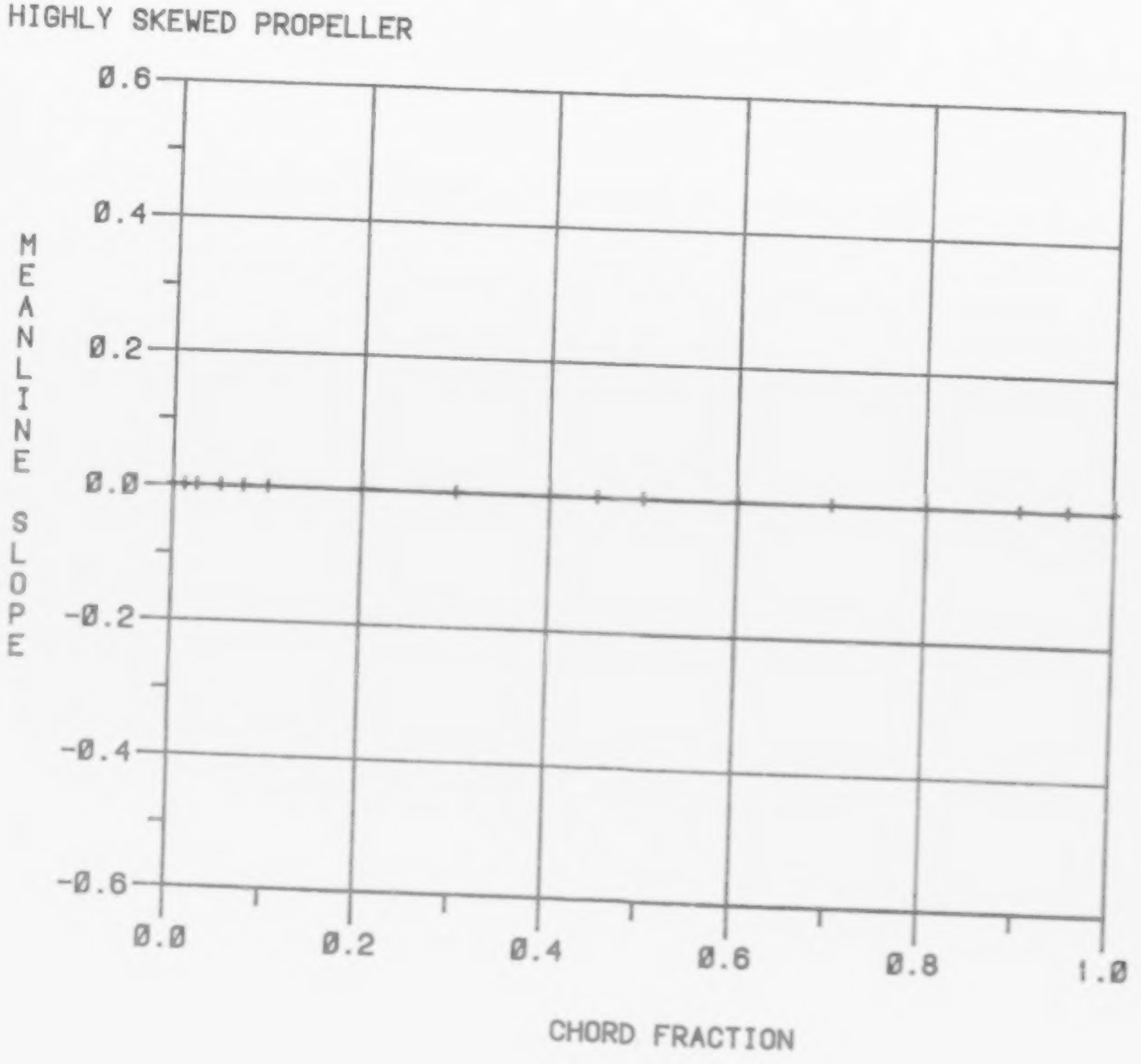


FIGURE M.22: Computer Plot of Mean Line Slope Versus Chord Fraction

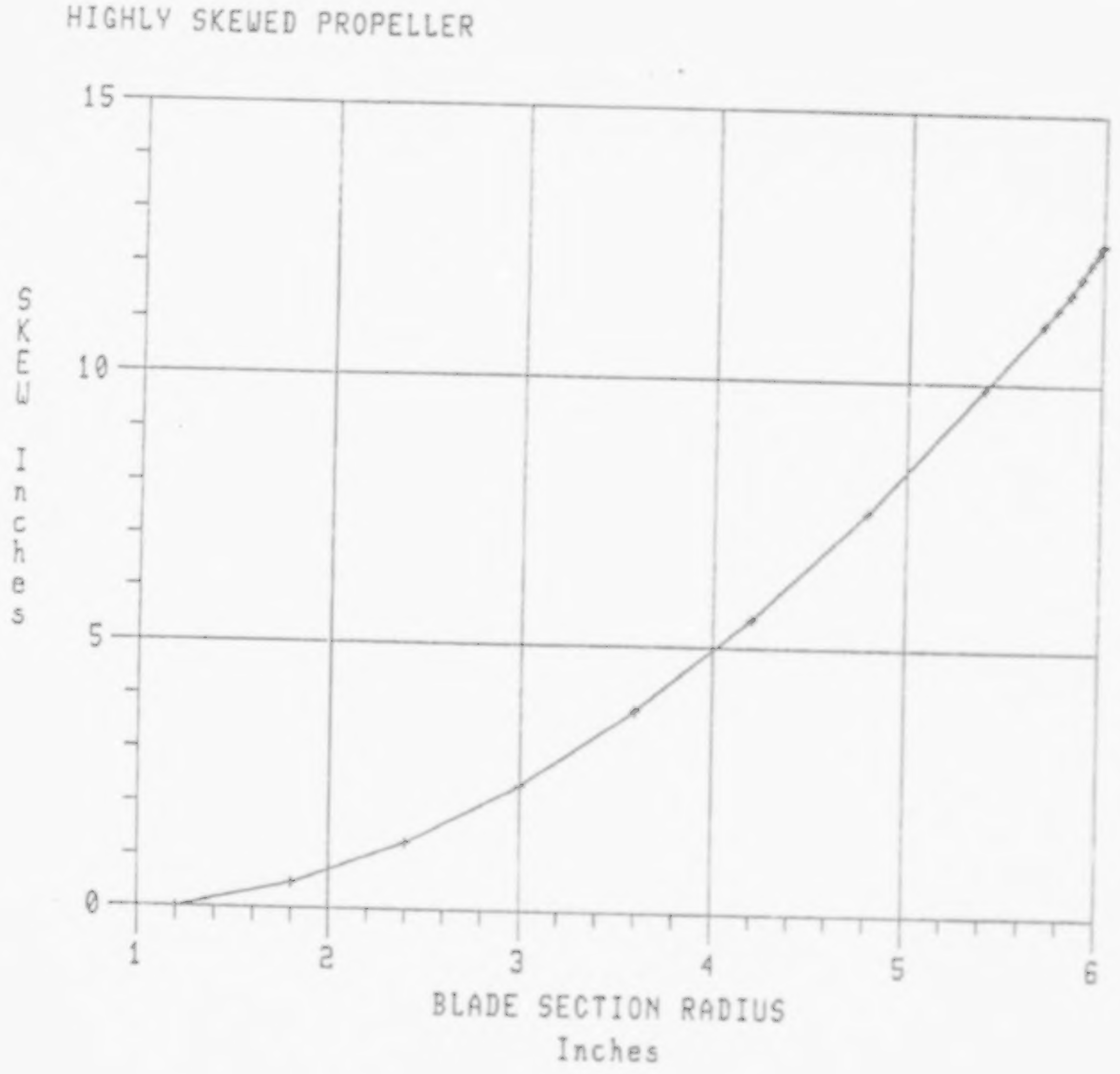


FIGURE M.23: Computer Plot of Skew Versus Blade Section Radius

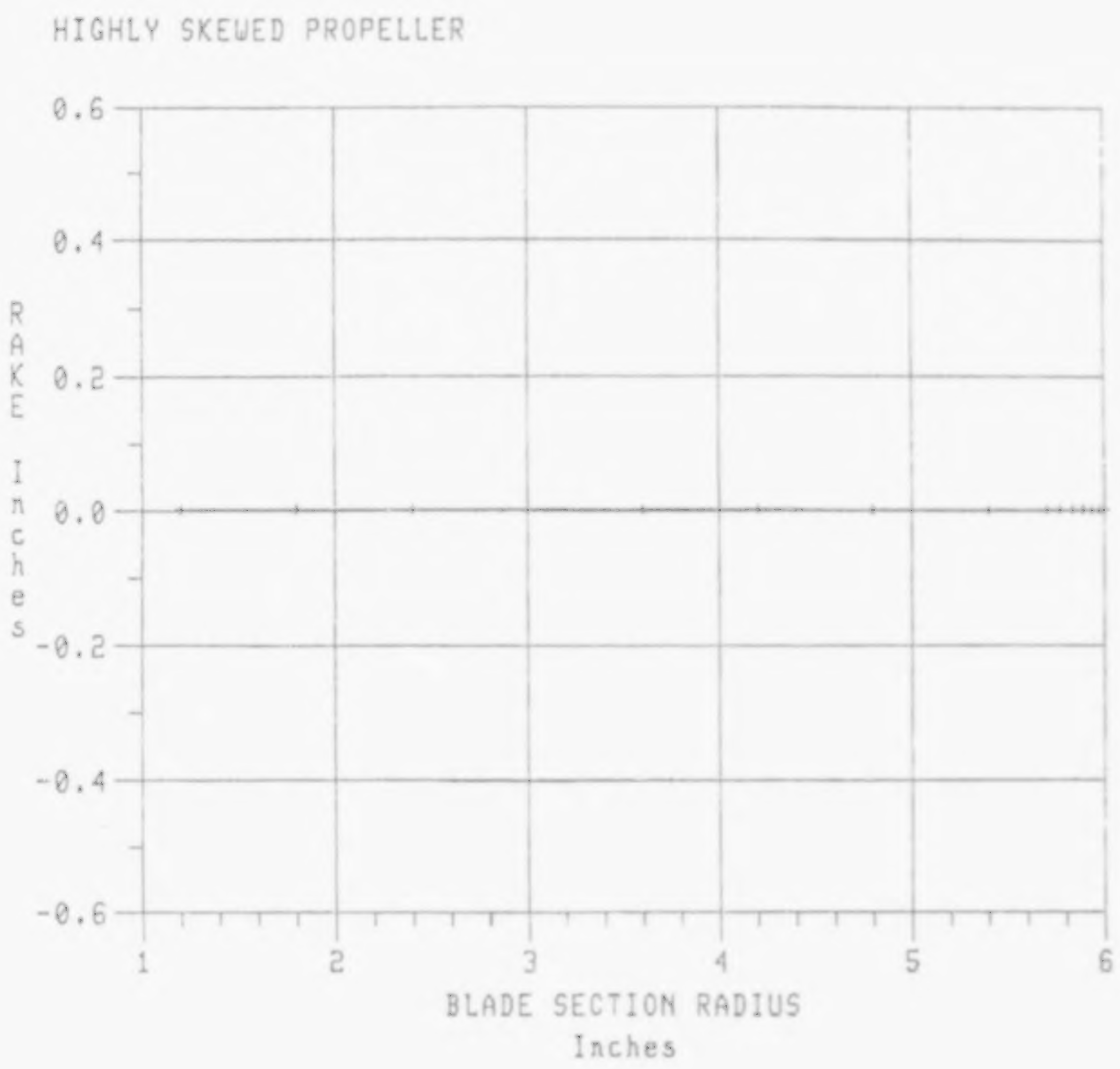


FIGURE M.24: Computer Plot of Rake Versus Blade Section Radius

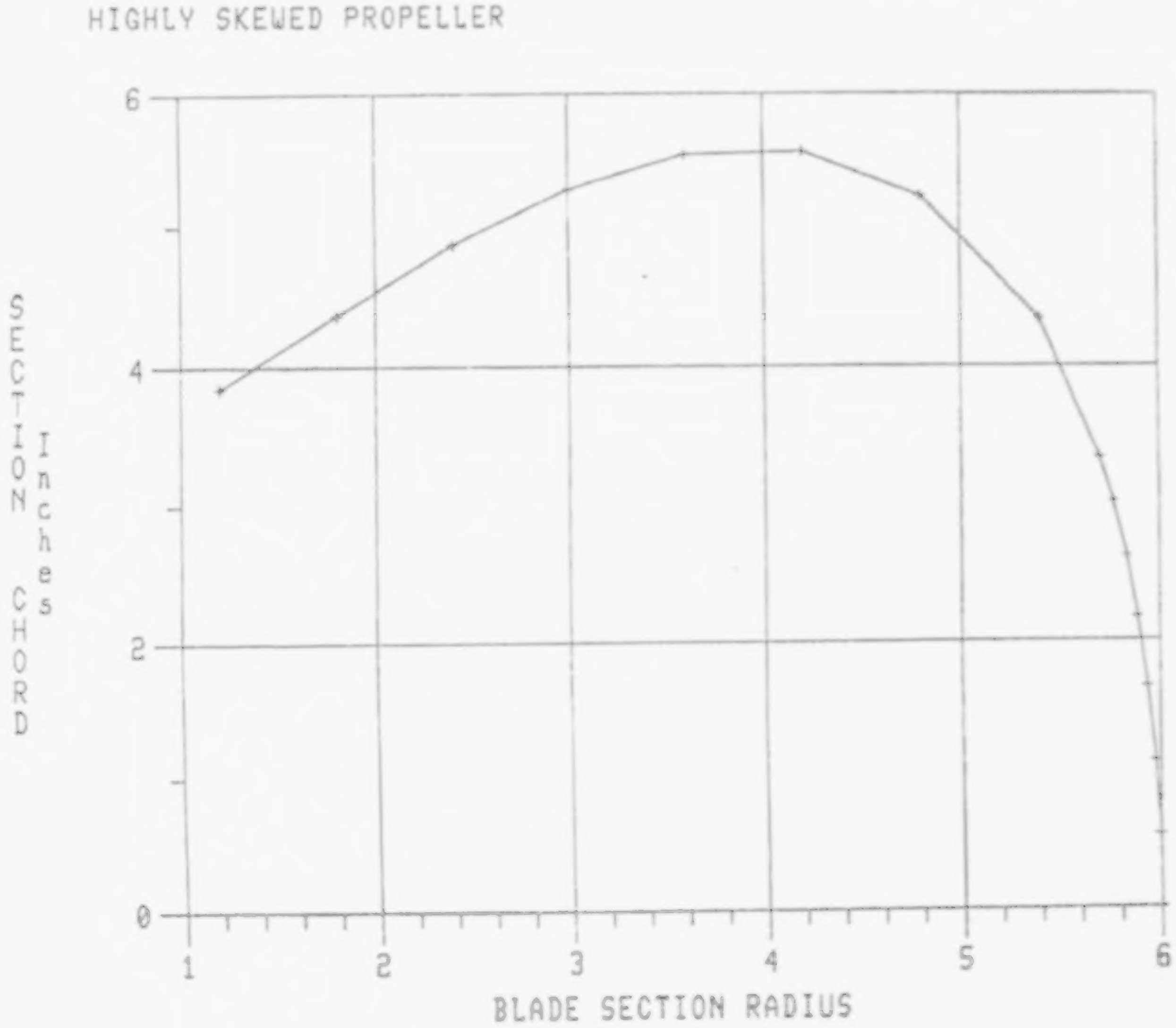


FIGURE M.25: Computer Plot of Section Chord Versus Blade Section Radius

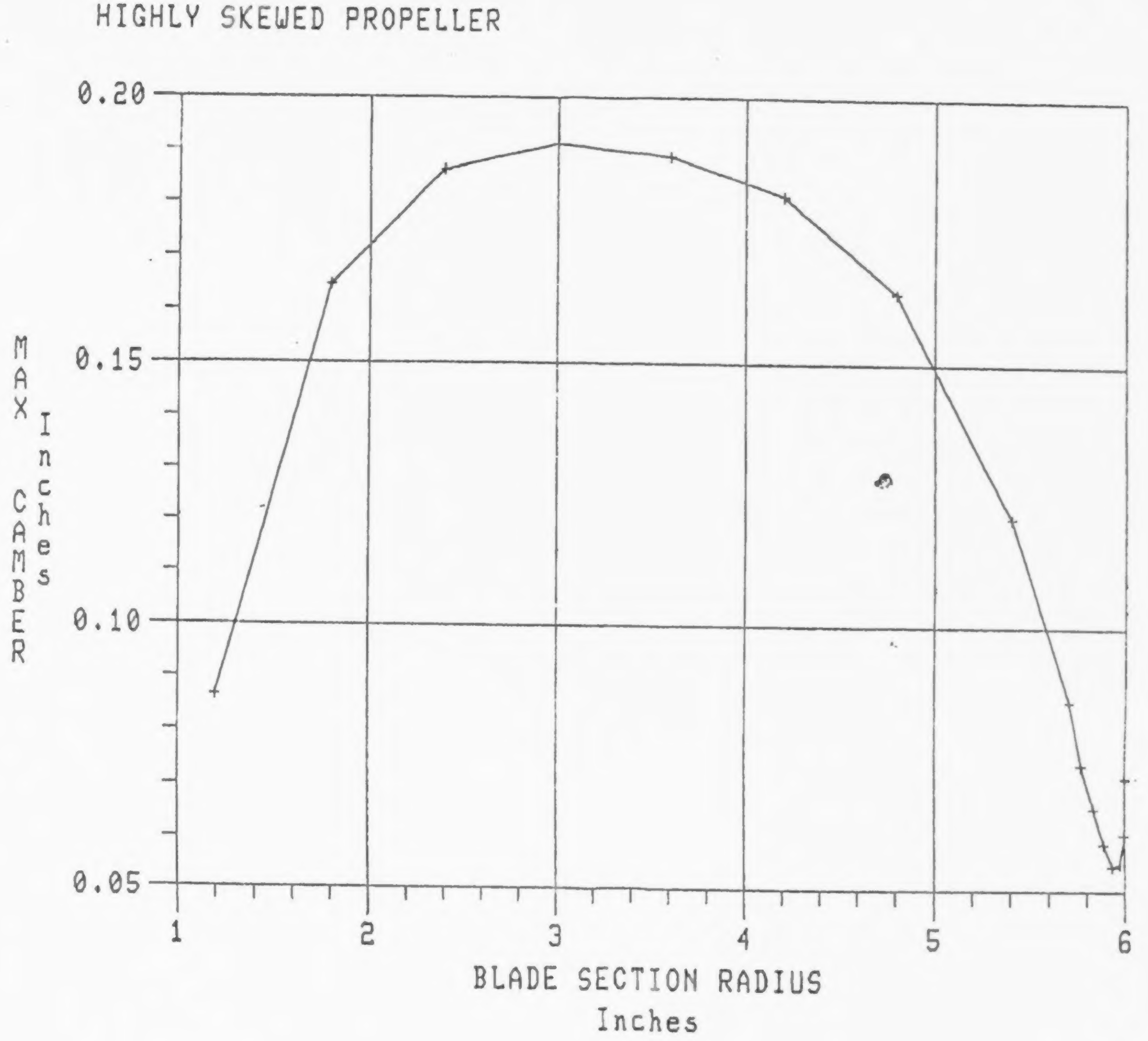


FIGURE M.26: Computer Plot of Maximum Camber Versus Blade Section Radius

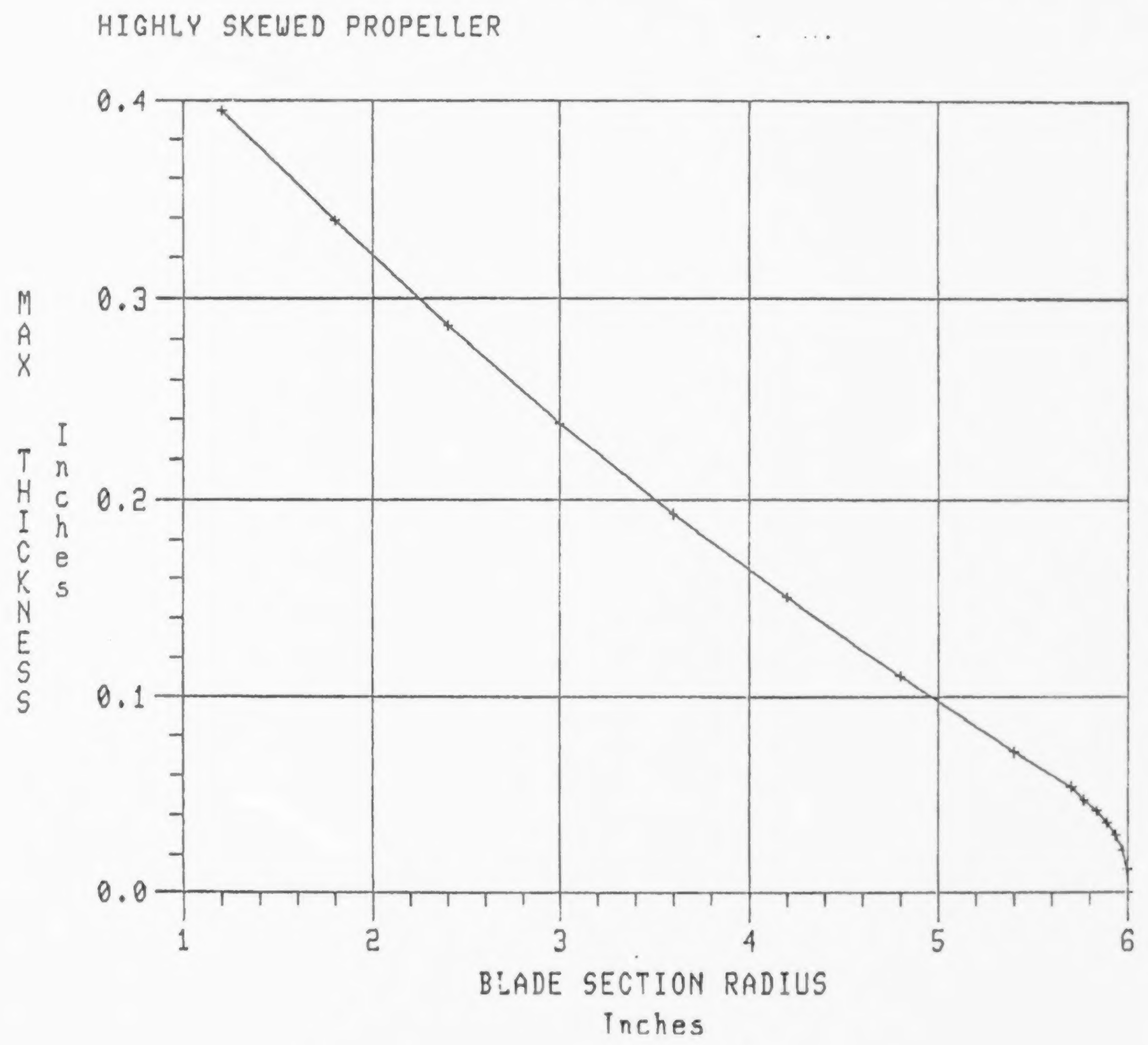


FIGURE M.27: Computer Plot of Maximum Thickness Versus Blade Section Radius

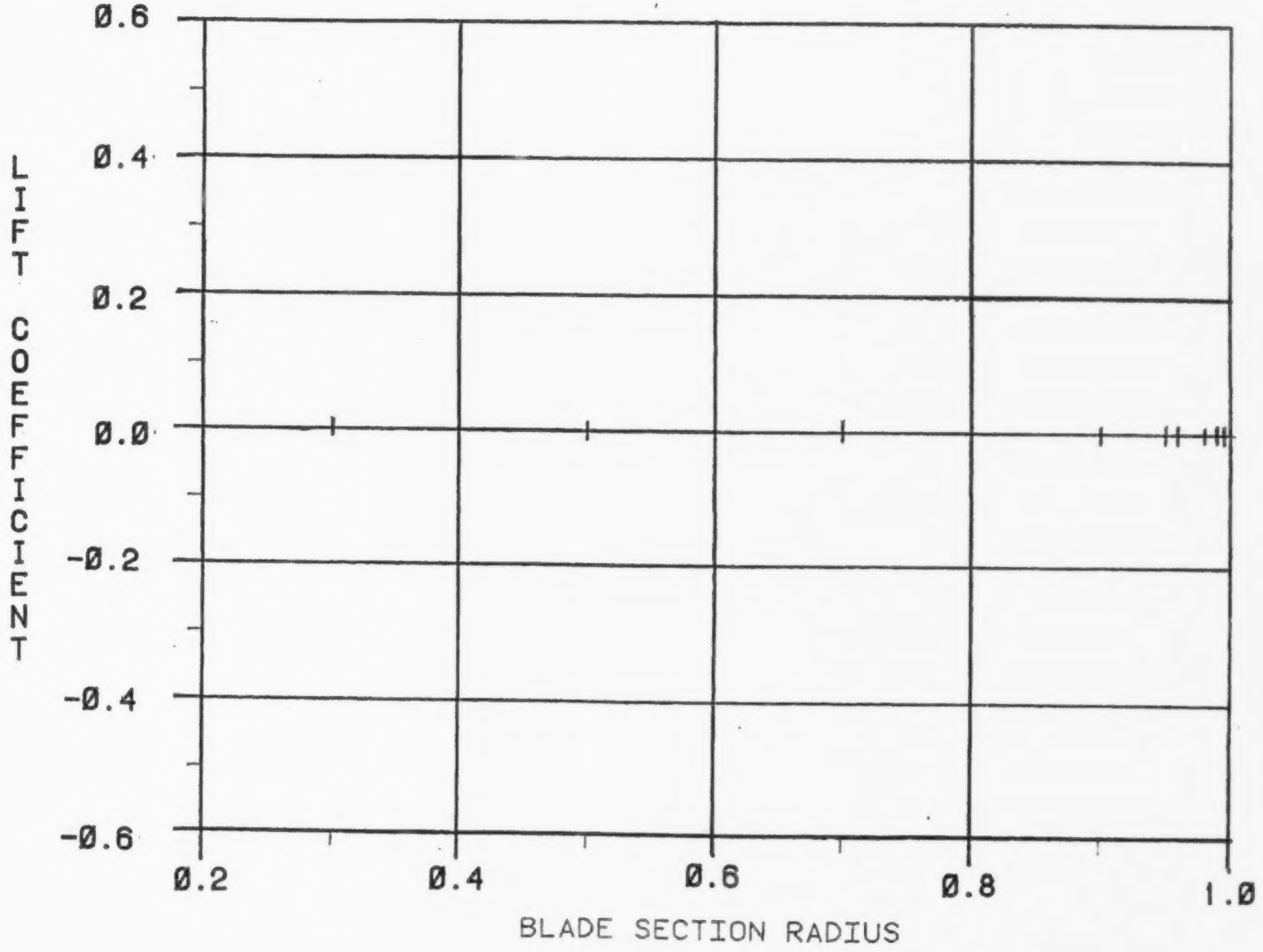


FIGURE M.28: Computer Plot of Lift Coefficient Versus Blade Section Radius

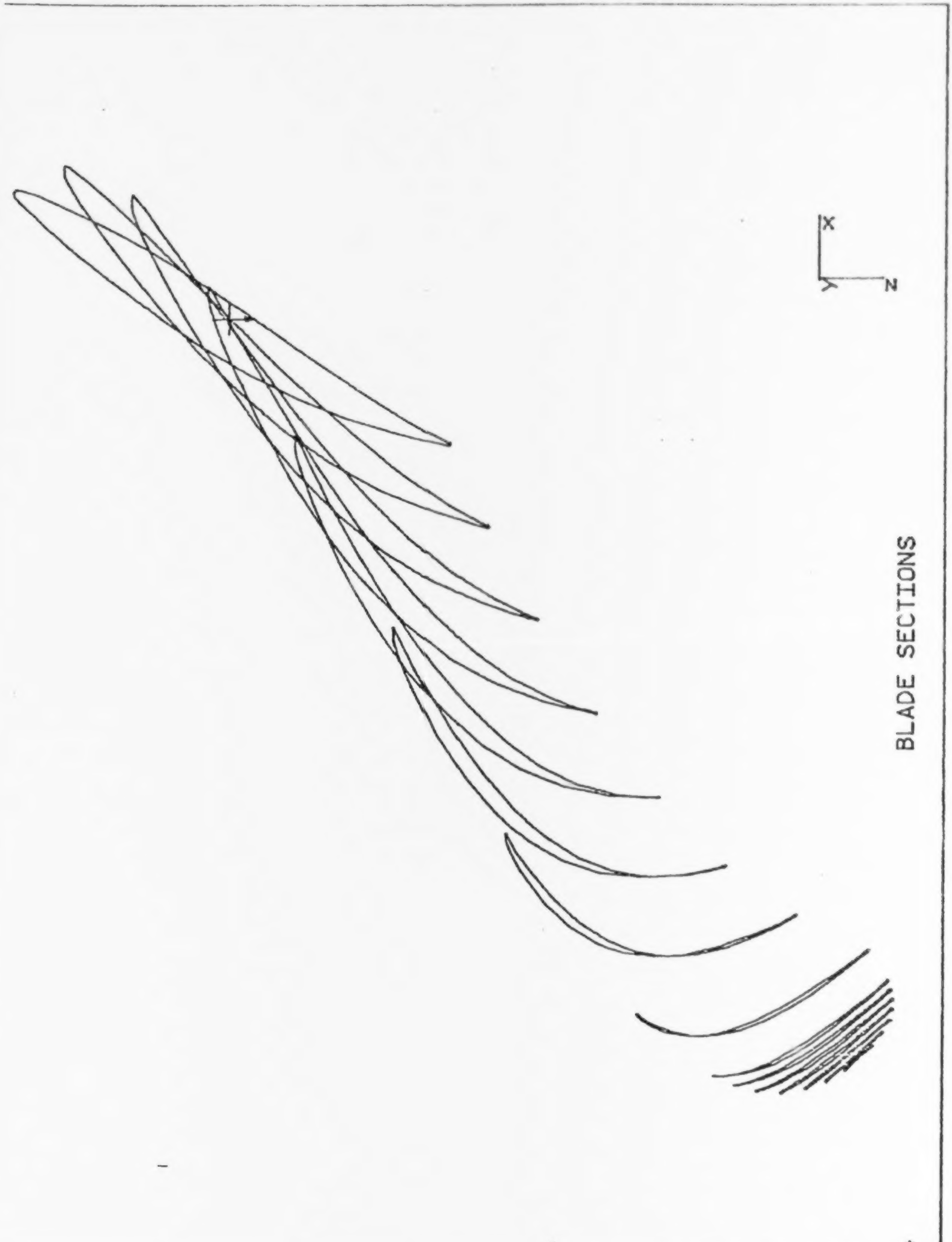

BLADE SECTIONS
X
Y
Z

FIGURE M.29: Computer Plot of Individual Blade Sections

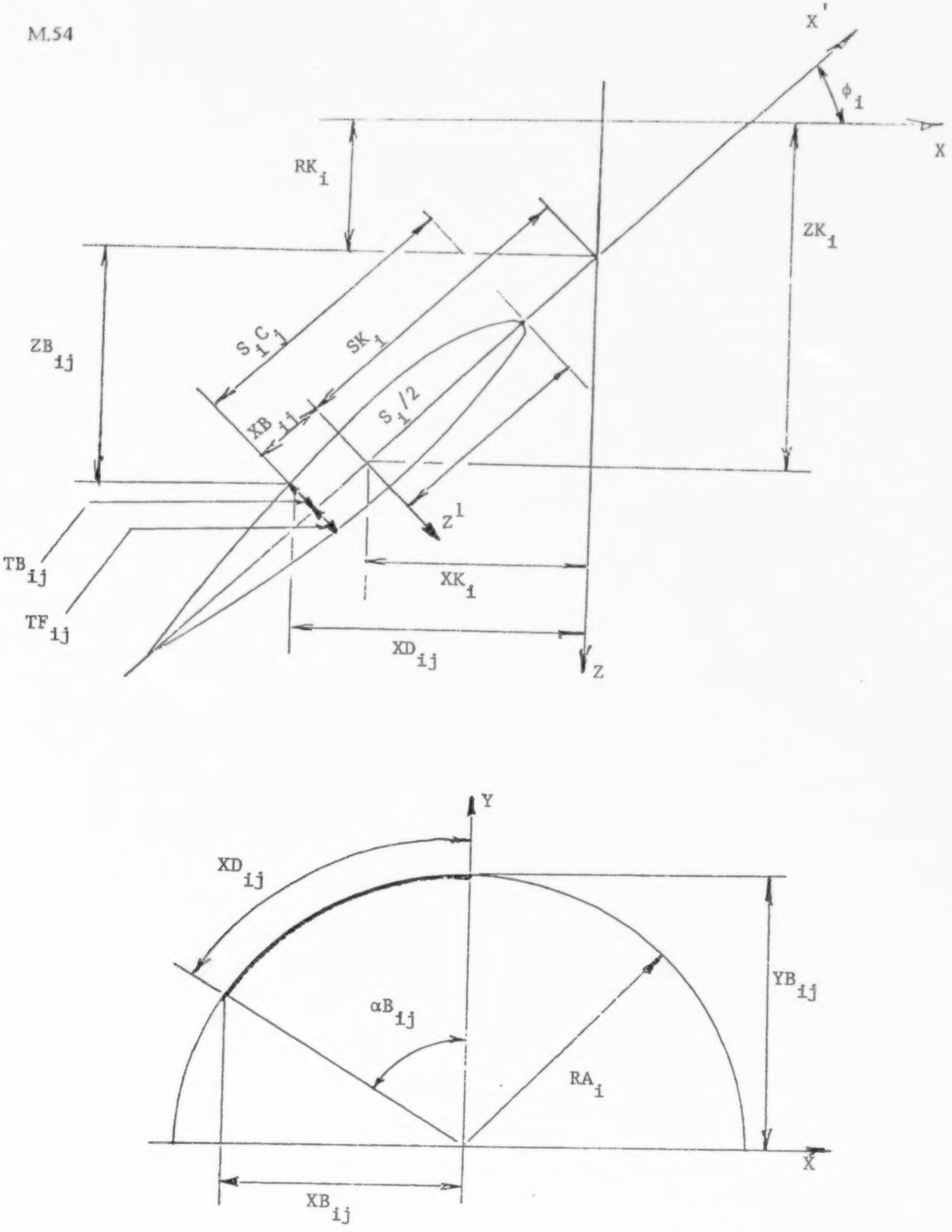

X'
φ i
X
RK i
ZK i
ZB ij
S i C i
SK i
S i /2
XB ij
Z 1
TB ij
TF ij
XK i
XD ij
Z
Y
XD ij
αB ij
YB ij
RA i
X
XB ij

FIGURE M.30: Generation of Spatial XYZ Coordinates

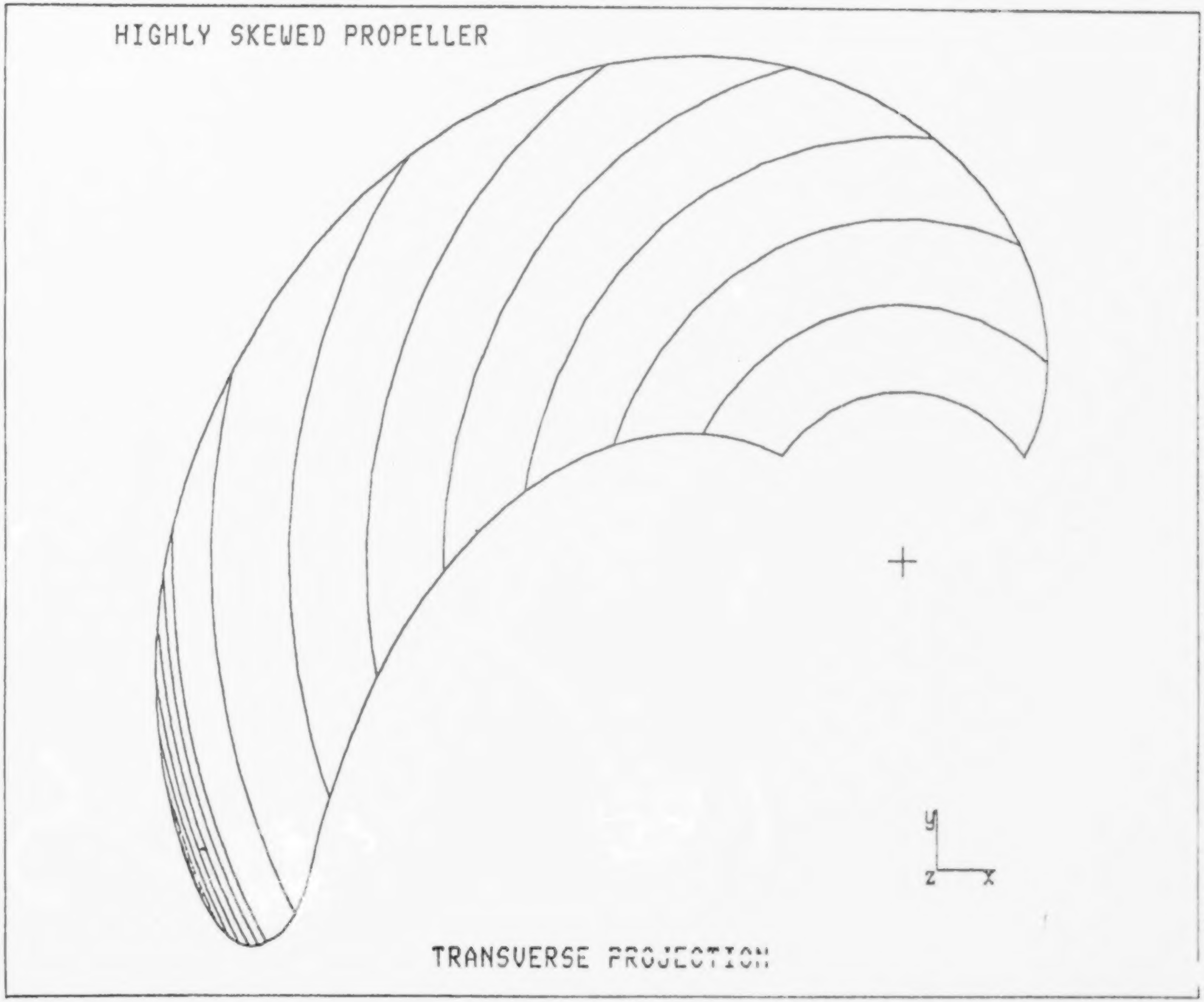

HIGHLY SKEWED PROPELLER
y
z
x
TRANSVERSE PROJECTION

FIGURE M.31: Computer Plot of Transverse View

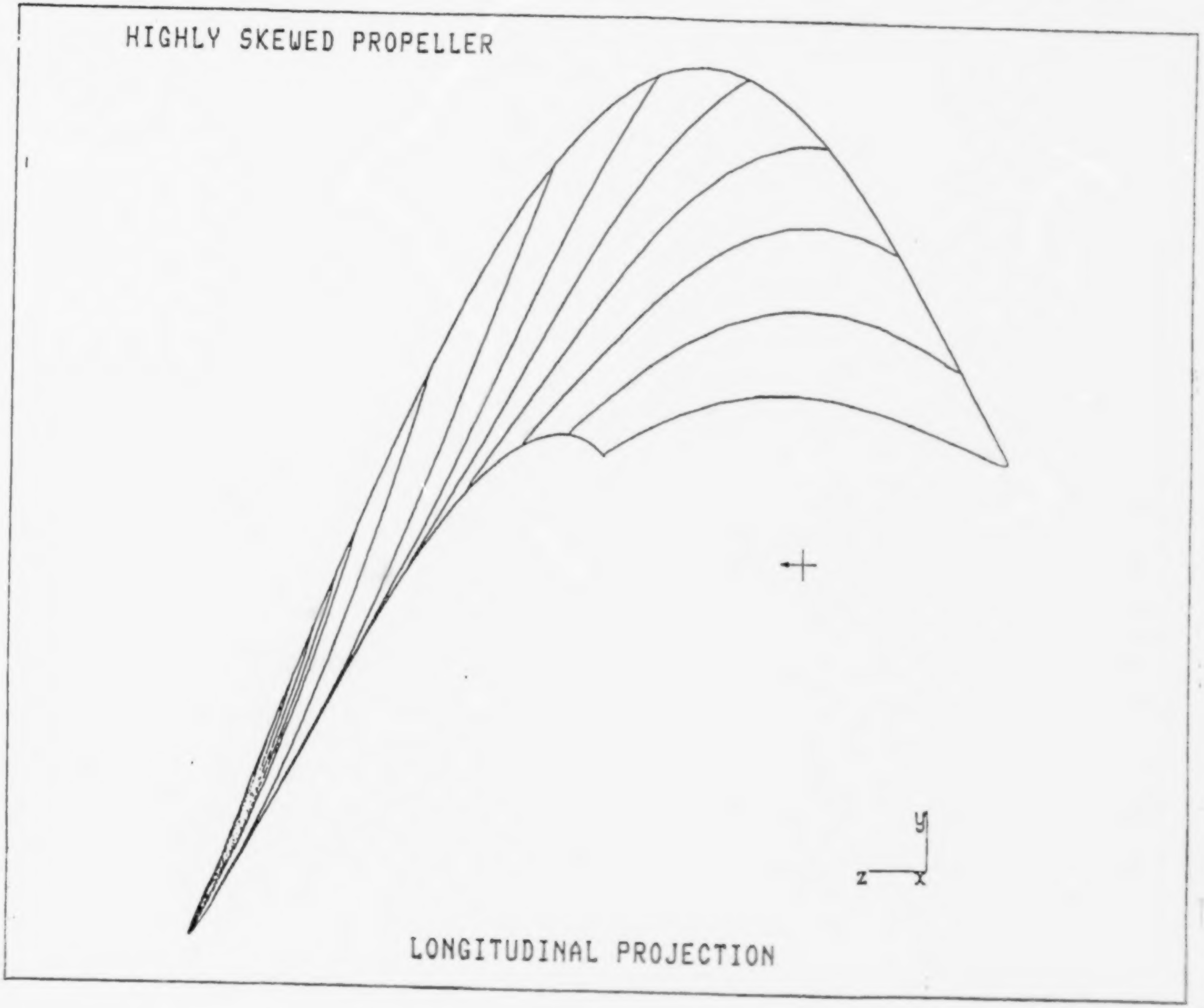

HIGHLY SKEWED PROPELLER
y
z
x
LONGITUDINAL PROJECTION

FIGURE M.32: Computer Plot of Longitudinal View

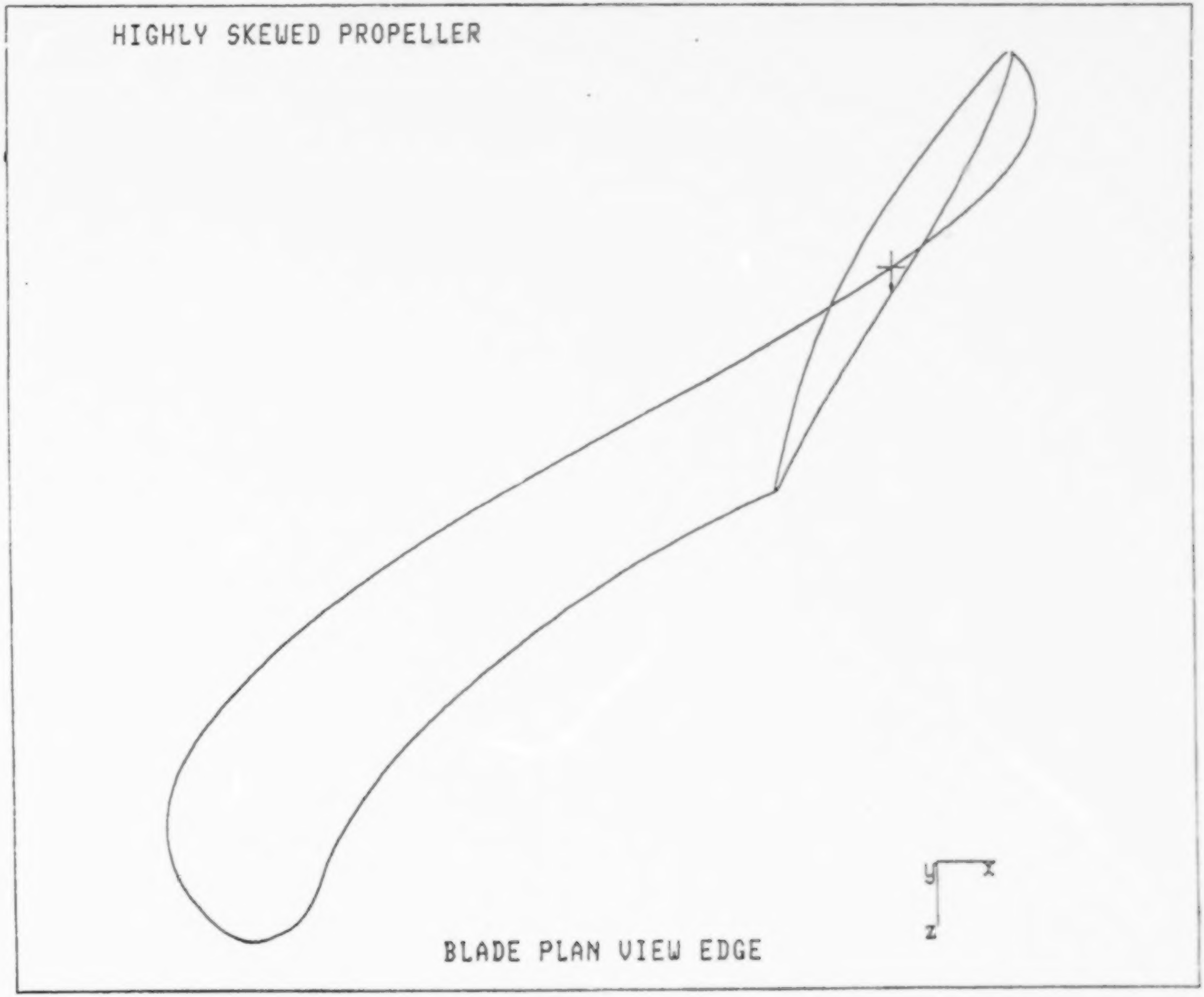

HIGHLY SKEWED PROPELLER
BLADE PLAN VIEW EDGE
y
x
z

FIGURE M.33: Computer Plot of Plan View

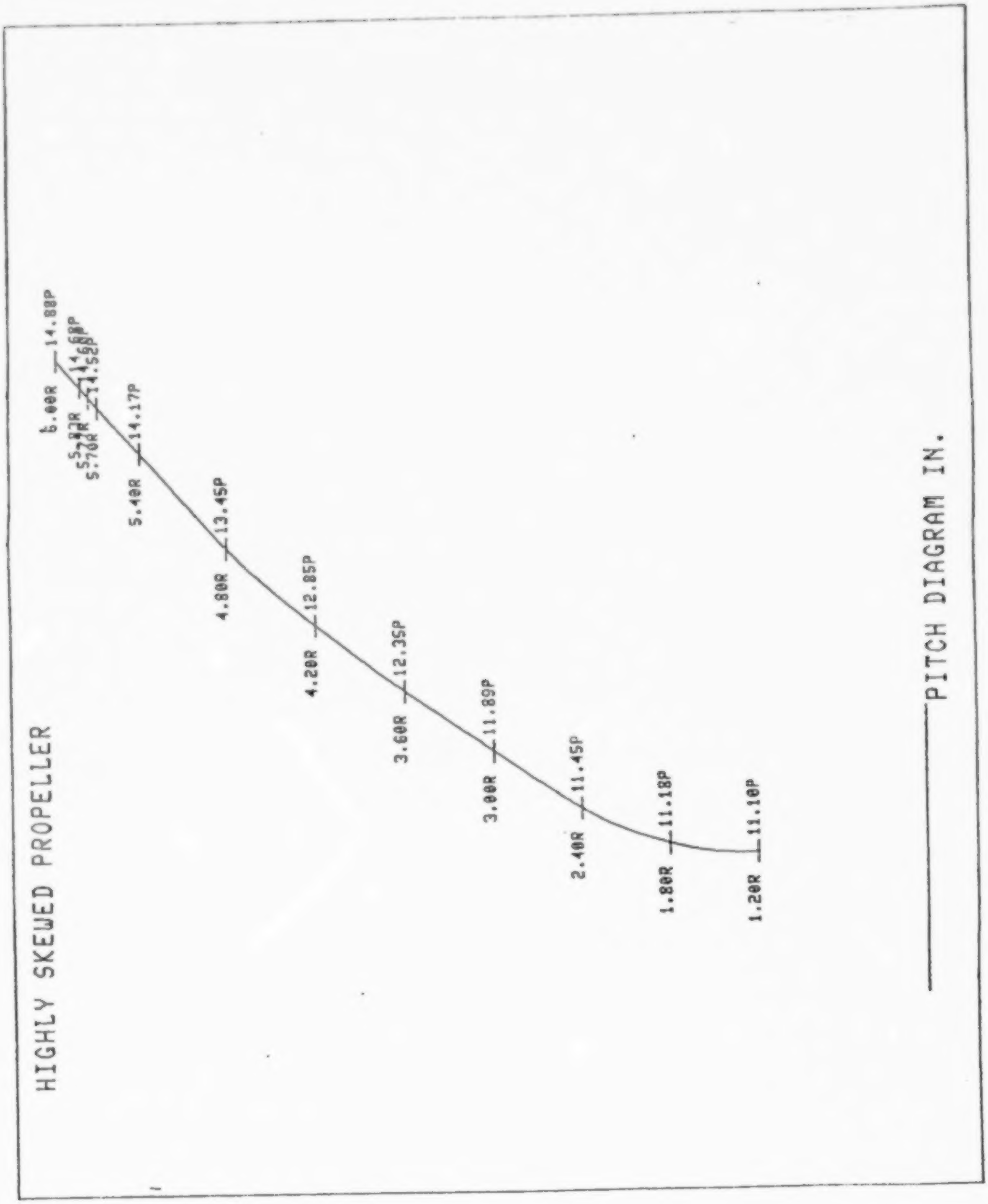


FIGURE M.34: Computer Plot of Pitch Diagram

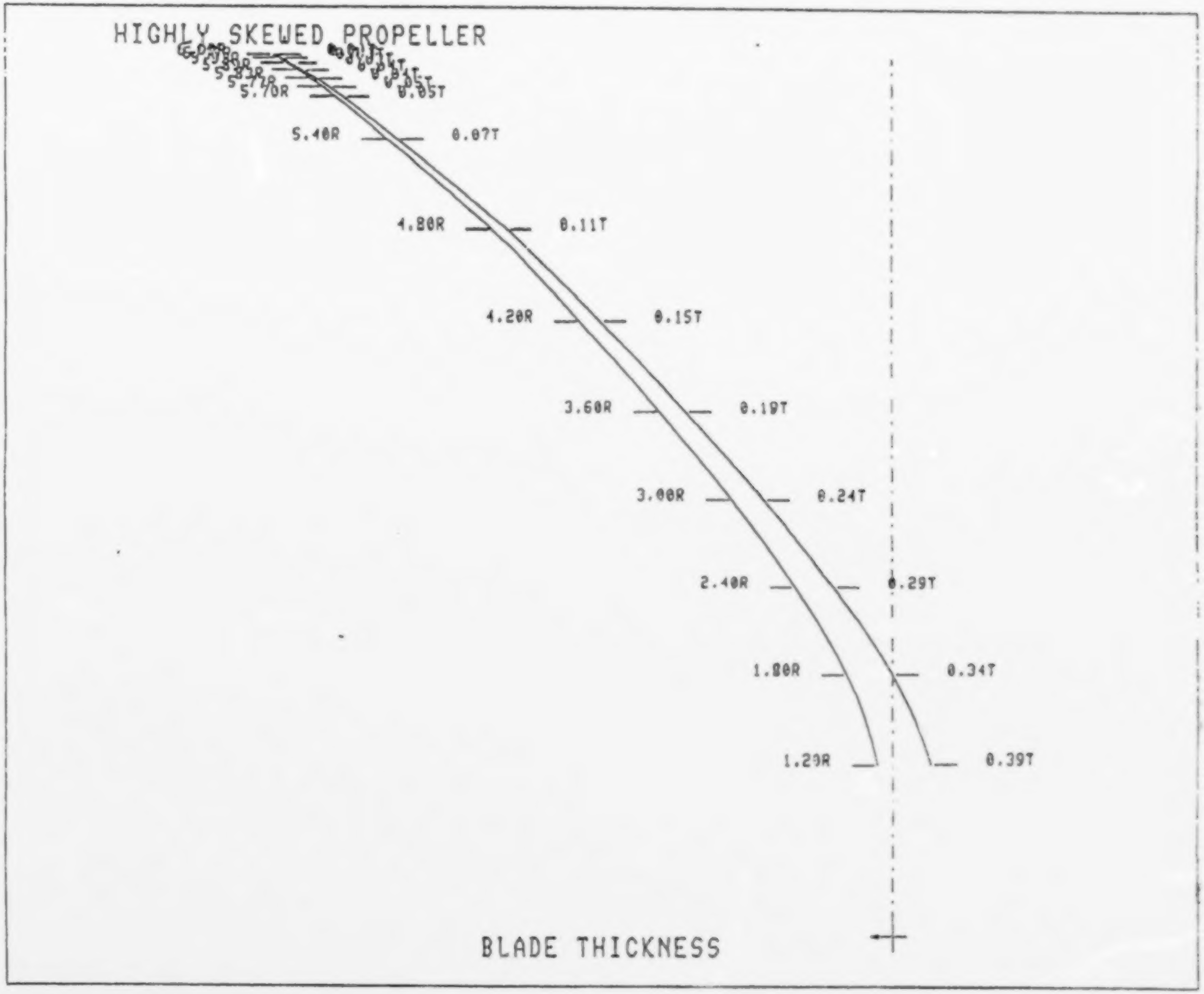


FIGURE M.35: Computer Plot of Blade Thickness and Rake Diagram

APPENDIX N

GENERATION OF AIR CHANNEL MODELS

PVCHL1

The subprogram PVCHL1 generates an air channel model and creates a PREFX.CHL file. This file contains all the data necessary for a VAST analysis and should be copied to a new PREFX1.USE file to perform detailed analysis of the air channel model. The program uses information generated from a previous analysis of a full propeller blade in which the effects of the air channel have been ignored. The set of data required to generate the air channel model is provided under the header "ACHANN" in the PREFX.CDR file. However, interactive input is required to define the point on the blade leading edge where the air channel model is to be located. In the following the air channel model data is first described to aid in creating the appropriate data in the card reader file. Then the terminal responses and data required to define the position of the air channel model are discussed. Terminal responses are in upper case while comments are provided in lower case.

Card 1 (2E10.3)

D = Depth of the air channel model perpendicular to the leading edge (length).

L = Length of the air channel model along the leading edge (length) (see Figure N.1).

Card 2 (6E10.3)

D1, α, A1, B1, C1, E1 are the air channel geometric specification parameters (see Figure N.2).

Note: α is in degrees and other quantities are in length units.

Card 3 (I5)

NEDGE = Number of layers of elements along leading edge.

Card 4 (I5)

IRESP = Parameter for specifying air channel pressure.
= 0, for no air channel pressure.
= 1, for air channel pressure to be provided.

Omit CARDS 5 and 6 if IPRESP = 0.

Card 5 (I5)

NTIME = Number of steps for which air channel pressure is provided.

Note: For static analysis set NTIME to 1. For dynamic analysis set NTIME to the same value as that used to describe the dynamic load in the original blade only analysis.

Card 6 (8F10.4)

ACHPR(1) = Air channel pressure at each time step.

ACHPR(NTIME)

Omit CARD 7 for static analysis (NTIME=1).

Card 7 (I5)

NNDP = 0, for dynamic results to be printed at all nodes.
= 1, for dynamic results to be printed at default nodes computed by PVAST.

Terminal Responses:

(1) SUBPROGRAM PVCHL1 PROGRAM TO GENERATE AIR CHANNEL F.E. MODELS FROM PROPELLER BLADE ANALYSIS

(2) SELECT A POINT ON THE LEADING EDGE WHERE THE REFINED AIR CHANNEL IS TO BE GENERATED AS FOLLOWS:

1. ENTER 1 to PLOT BLADE
2. MOVE CURSOR TO POINT ON BLADE LEADING EDGE
3. ENTER ANY CHARACTER TO DETERMINE LOCATION
4. PRESS RETURN TO PROCEED

Enter 1 (else the above prompt repeats until 1 is entered) to put the program in graphics mode.

The outline of the original blade now appears on the screen followed by the cursor. Move the cursor to the desired point at which the air channel model is to be located. Enter any character to mark the selected point. Press return to continue.

(3) This prompt and step 4 are skipped if location of point is adequate.

THE POINT IS TOO CLOSE TO THE END OF THE LAST ELEMENT AT THE TIP OR ROOT ON THE LEADING EDGE.

ENTER: 1 TO SELECT ANOTHER POINT CLOSER TO THE CENTER OF THE LEADING EDGE

2 TO REDUCE THE LENGTH OF THE CHANNEL MODEL ALONG THE LEADING EDGE

This prompt appears if the point selected in step (2) above is too close to the root or tip such that the channel model cannot be fitted without it overhanging. Review the air channel model by either selecting another point (enter 1) or reducing the length of air channel model (enter 2).

If 1 is entered step (2) is repeated and program continues.

If 2 is entered step (4) is activated and a new length can now be entered and the program again checks the adequacy of the location of the point.

(4) ENTER THE LENGTH OF THE CHANNEL MODEL ALONG THE LEADING EDGE:

Enter a new channel length (see Figure N.1). The program now uses this new length in the computations instead of the air channel length provided in the PREFX.CDR file.

(5) The program then computes and displays the fraction radius of the points selected and prompts:

IS THIS CLOSE ENOUGH TO THE DESIRED r/R (0=NO, 1=YES)?:

Respond appropriately. If a 0 is entered the program goes back to step (2). If 1 is entered program continues and generates the air channel model.

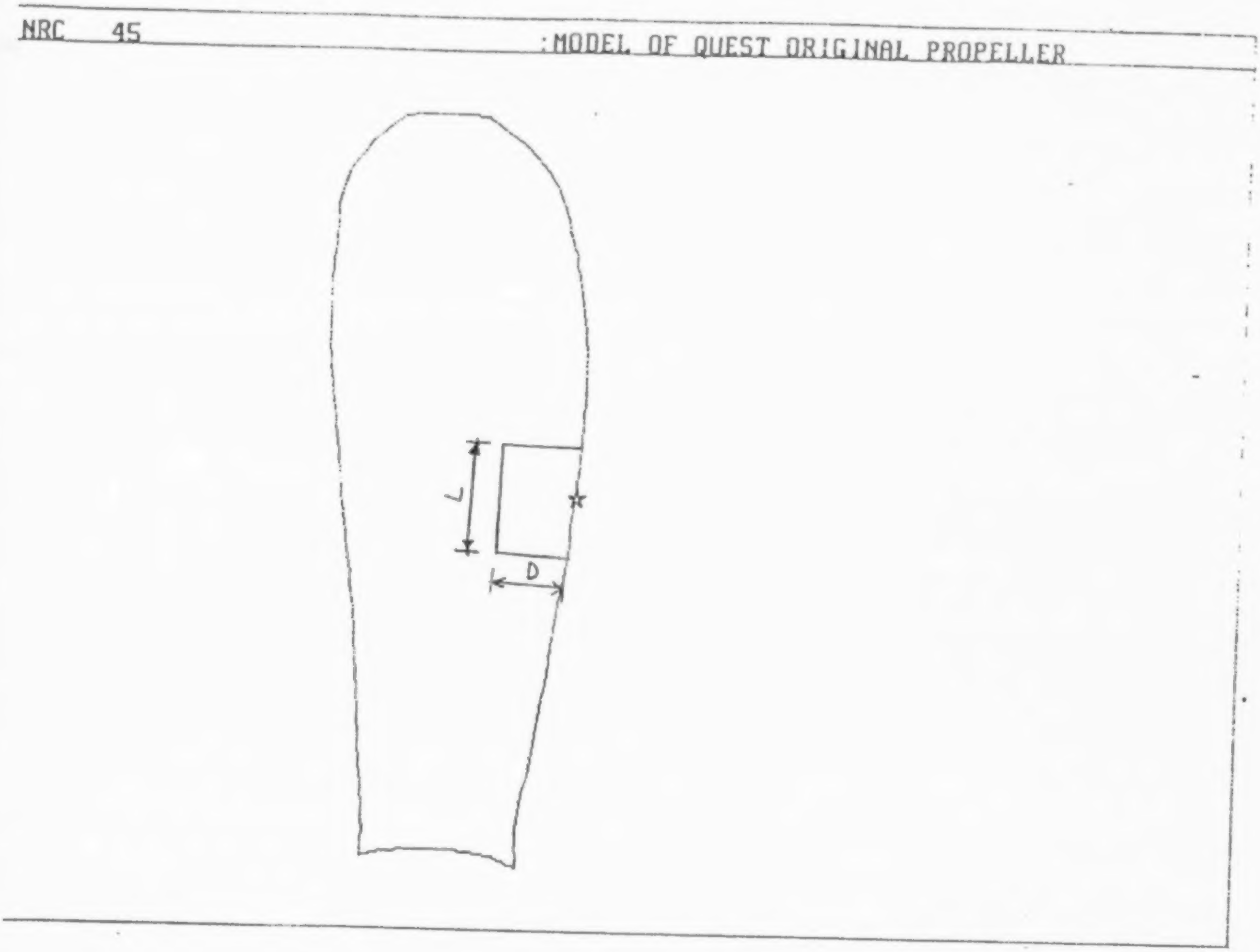
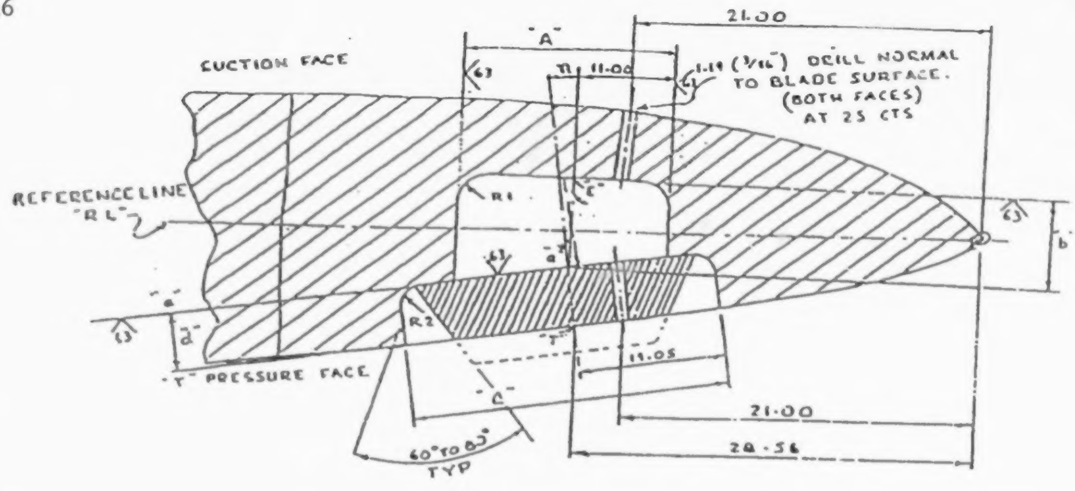
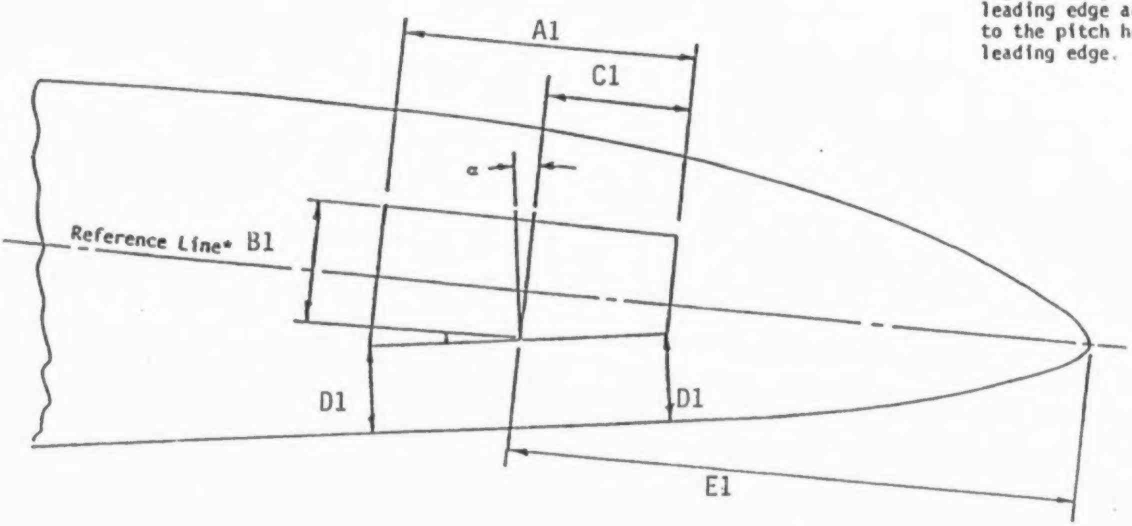

NRC 45
:MODEL OF QUEST ORIGINAL PROPELLER
L
D
☆

FIGURE N.1: Full Blade Model


21.00
"A"
SUCTION FACE
63
R1, 11.00
1.19 (3/16") DRILL NORMAL TO BLADE SURFACE. (BOTH FACES) AT 25 CTS
REFERENCE LINE "RL"
R1
"E"
63
"a"
63
"b"
"e"
63
"d"
R2
"T"
11.05
"T" PRESSURE FACE
"C"
21.00
60° TO 85° TYP
28.56

(a) Air channel fabrication details


A1
C1
α
Reference Line* B1
D1
D1
E1

*Reference Line: Line of intersection of a plane normal to the tangent at the leading edge and a plane containing the tangent to the leading edge and tangent to the pitch helix at the leading edge.

(b) Air channel geometric specifications

FIGURE N.2: Propeller Blade Internal Air Channel Specifications

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

Martec Ltd.
Suite 400, 1888 Brunswick St., Hfx., N.S. B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable)

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

Propeller Vibration and Stress Analysis by Finite Element Methods, PVAST Version 4 User's Manual

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

Chernuka, M.W., Norwood, M.E., Koko, T.S.

5. DATE OF PUBLICATION (month and year of publication of document)

July 1992

a. NO OF PAGES (total containing information include Annexes, Appendices, etc).

267

6. NO. OF REFS (total cited in document)

14

6. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).

DREA Contractor Report

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include the address).

Defence Research Establishment Atlantic
Grove Street, Dartmouth, Nova Scotia B2Y 3Z7

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).

9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).

W7707-0-1091/01-OSC

10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document).

10b. OTHER DOCUMENT NOS. (Any other numbers which may be assigned this document either by the originator or by the sponsor).

DREA CR 92/454

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Distribution limited to defence departments and defence contractors; further distribution only as approved
( ) Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved
( ) Distribution limited to government departments and agencies; further distribution only as approved
( ) Distribution limited to defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected).

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

JGY A User's Manual for a computerized procedure for propeller vibrations and stress analysis by finite element methods is presented.

The manual provides a general description of the procedure, including the analysis capabilities, the numerical techniques used to formulate and solve the equations of motion, modelling techniques, and computer program organization and operation.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

propellers
finite element
strength
vibration
stress
fluid interaction